IN-39
7296
p 437

NASA Contractor Report 185298

BEST3D User's Manual

Boundary Element Solution Technology, 3-Dimensional Version 3.0

Computational Engineering Mechanics Laboratory
Department of Civil Engineering
State University of New York at Buffalo

United Technologies Corporation
Pratt & Whitney
Commercial Engineering
East Hartford, Connecticut

March 1991

Prepared for
Lewis Research Center
Under Contract NAS3-23697

NASA
National Aeronautics and
Space Administration

(NASA-CR-185298) BEST3D USER'S MANUAL: BOUNDARY ELEMENT SOLUTION TECHNOLOGY, 3-DIMENSIONAL VERSION 3.0 (State Univ. of New York) 437 p CSCL 20K
N91-20538
Unclas
G3/39 0007296

BEST3D
USER'S MANUAL

***B**oundary **E**lement **S**olution **T**echnology **3D**imensional*

Version 3.0
March 1991

P. K. Banerjee and R. B. Wilson

ACKNOWLEDGEMENTS

It should be emphasised that a large undertaking such as the development of **BEST3D** is possible only through the collective efforts of a large number of people. To Dr. Chris Chamis and Mr. Dale Hopkins of NASA Lewis Center, Cleveland, who had the vision to conceive this project, and to Dr. Edward Todd of Pratt & Whitney for inspired leadership we express our sincere gratitude. A very significant contribution to the development effort was made by David Snow of Pratt & Whitney, who had the major responsibility for the validation examples and Nancy Miller who developed substantial part of the utility routines.

To Drs. T. G. Davies and G. D. Manolis of the Department of Civil Engineering, State University of New York at Buffalo, who carried out the early part of the investigations for the inelastic and dynamic analyses, respectively, we express our deepest appreciation for their work. A number of former and present graduate students (at SUNY, Buffalo) of exceptional academic ability and programming skill contributed substantially to the development of **BEST3D.** In particular we are indebted to Drs. S.T. Raveendra and D. Henry for their major contributions in inelastic analyses, to Dr. N. B. Yousif for her contributions in inelastic constitutive equations, to Dr. R. Sen for the early part of the work on the transient dynamic analyses, to Drs. S. Ahmad and Wang for their major contributions in periodic dynamic, transient time domain dynamic analysis algorithms, and to Dr. G. Dargush for his major contributions in thermal problems.

TABLE OF CONTENTS

1.0 INTRODUCTION

BEST3D (Boundary Element Solution Technology in Three Dimensions) is an advanced engineering analysis system based upon the the boundary element method (**BEM**). For many problems, BEM offers significant advantages in accuracy, efficiency, and ease of use over the more familiar finite element method.

The overall **BEST3D** system has two primary objectives :

. Provide new and/or improved capability for the engineering analyst.

. Provide a complementary analysis technique for the verification of finite element or experimental results.

Consequently, **BEST3D** has been designed to solve large realistic problems from a wide range of engineering disciplines, utilizing state-of-the-art boundary element technology. Table 1.1 highlights some of the general features of the code, while a brief summary of the capabilities of **BEST3D** is presented by analysis type in Table 1.2.

The primary objective of this User's Manual is to provide an overview of all **BEST3D** capabilities, along with detailed descriptions of the input data requirements. As a result, the present manual does not provide a thorough development of the theoretical basis for the various types of analysis, nor does it contain a comprehensive collection of realistic example problems. However, sufficient material is included so that, after receiving this manual an engineering analyst should be able to prepare **BEST3D** input data sets.

In the next chapter, a brief review of the theoretical background is presented for each analysis category. In most cases, only sketchy details are provided, however references are cited to assist the interested reader. Then, Chapter 3 discusses the key aspects of the numerical implementation, while Chapter 4 provides a tutorial for the beginning **BEST3D** user. The heart of the manual, however, is in Chapter 5, where a complete description of all data input items is provided. Within this chapter, the individual entries are grouped on a functional basis for a more coherent presentation. Chapter 6 includes sample problems for all the major analysis types. The complete input data sets are heavily commented, and thus should be of considerable assistance to the novice. Actual extracts from various analysis output options are also provided. Chapter 7 includes capsules of a number of realistic engineering analysis problems that have been solved using **BEST3D** . This chapter is primarily descriptive in nature and is intended merely to illustrate the level of engineering analysis that is possible within the present **BEST3D** system. Finally, all pertinent references are listed in Chapter 8.

TABLE 1.1
GENERAL FEATURES OF BEST3D

- Three dimensional analysis
- Conforming element approach to provide inter-element continuity of the field variables, along with efficient solutions
- Substructuring to permit multiple materials and more efficient solutions
- Automatic adaptive numerical integration schemes
- Cyclic and planar symmetry
- Local or global boundary condition specification
- Sliding, spring and resistance-type interfaces
- Exterior domains and infinite elements
- Block banded solver routines based upon LINPACK
- Generalized eigenvalue extraction routines based upon EISPACK
- Restart capability to reuse existing integration coefficients
- Free-format, keyword-driven input
- Automatic input data error checking

TABLE 1.2
ENGINEERING ANALYSIS BY BEST3D

A. Elastic Analysis

- Stress analysis and free-vibration analysis
- Analytic representations of holes in 3-D solids with a minimum of discretization
- Isotropic, as well as anisotropic media
- Centrifugal and self-weight body forces
- Spring loaded or sliding interfaces
- Embedded inserts for analysis of 3-D composites.

B. Dynamic Analysis

- Forced periodic response
- Forced transient response in time domain
- Free-vibration analysis

C. Thermal Stress Analysis

- Steady state linear thermal stress analysis
- Nonlinear (thermoplastic) stress analysis
- Nonlinear thermoviscoplastic stress analysis
- Analytic representations of holes in 3-D solids with a minimum of discretization
- A large family of constitutive models for nonlinear analysis
- Hotspots and discontinuities

TABLE 1.2 (continued)
ENGINEERING ANALYSIS BY BEST3D

D. Heat Transfer Analysis

- Steady state heat transfer
- Transient heat transfer
- Analytic representations of cooling holes in 3-D with a minimum of discretization
- Thermal resistance across subregions

E. Nonlinear Stress Analysis

- Iterative elastoplasticity and viscoplasticity
- Direct variable stiffness type elastoplasticity
- Cyclic elastoplasticity
- Inelastic fracture mechanics analysis
- Thermoplasticity and thermoviscoplasticity

2.0 THEORETICAL BACKGROUND

The mathematical background of the boundary element method has been known for nearly 100 years. Indeed some of the boundary integral formulations for elastic, elastodynamic, wave and potential flow equations have existed in the literature for, at least, 50 years. With the emergence of digital computers the method had begun to gain popularity as the 'boundary integral equation method', 'panel method', or 'integral equation method' during the sixties. The name was changed to the 'Boundary Element Method' by Banerjee and Butterfield [1] in 1975, so as to make it more appealing to the engineering analysis community. Since then a number of textbooks and advanced level monographs have appeared [1- 13] which give a very comprehensive and thorough account of the existing literature on the method.

In this section, the different types of analyses incorporated in **BEST3D** are briefly outlined.

2.1	QUASI-STATIC ELASTIC ANALYSIS

It is well known that, for a homogeneous, elastic material, there exist point force solutions. These solutions provide the displacements at any point in an infinite body due to the application of a unit load at any other point in the body. Closed form solutions are available for isotropic materials (the Kelvin solution) and for transversely isotropic materials. Differentiation and the use of the appropriate elastic stress- strain relationship allow calculation of the stresses, strains and tractions due to the point load.

The Reciprocal Work Theorem

$$\int_S t_i(x)u_i^*(x)ds + \int_V f_i(x)u_i^*(x)dv = \int_S t_i^*(x)u_i(x)ds + \int_V f_i^*(x)u_i(x)dv \tag{2.1.1}$$

is the integral identity satisfied for any two equilibrium states of tractions (t), displacements (u), and body forces (f) existing in a solid occupying the volume V with surface S. Using the point load solution as one of the two equilibrium states and the solution of the desired boundary value problem as the second, it is possible to obtain [2]:

$$u_j(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds + \int_V G_{ij}(x,\xi)f_i(x)dv \tag{2.1.2}$$

the interior displacement identity expressing the displacement at any interior point in terms of surface integrals of the displacements and tractions and volume integrals of mechanical body forces, thermal strains and initial (inelastic) stresses. Since the point load solution is singular at the load point, special attention is required to limiting operations during the integration.

It is possible to take the load point to the boundary of the body, obtaining:

$$C_{ij}u_i(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]ds + \int_V G_{ij}(x,\xi)f_i(x)dv \tag{2.1.3}$$

a constraint equation relating the known and unknown displacements and tractions on the surface of the body. Further, differentiation and the use of the elastic stress-strain relations allow the evaluation of strains and stresses at arbitrary points within the body. For example,

$$\sigma_{ij}(\xi) = \int_S [G^\sigma_{kij}(x,\xi)t_k - F^\sigma_{kij}(x,\xi)u_k(x)]ds + \int_V G^\sigma_{kij}(x,\xi)f_k(x)dv \tag{2.1.4}$$

can be written for interior stresses. In the simplest case, a homogeneous elastic problem without thermal or mechanical body forces, all of the volume integrals vanish and (2.1.3) allows the solution of the boundary value problem entirely in terms of boundary geometry and the values of displacement and traction on the boundary. No reference need be made

to the interior of the body unless calculations of interior values of stress or displacement are desired after the boundary solution has been completed. The boundary element method was first developed for this class of problems, and was referred to in the literature as the BIE (Boundary Integral Equation) method.

The volume integrals corresponding to any steady state thermal loading or conservative mechanical body force can be converted to surface integrals, and such problems can thus be handled within the classical BIE formulation [14,15], although this is not the approach adopted in **BEST3D**. Rather, all of these effects on linear analyses have been incorporated using particular integrals where no surface integrations are required.

This new approach, based on particular integrals, is adopted in **BEST3D** for the treatment of centrifugal body forces. The method is unique since no volume integration or additional surface integration is required as in alternative formulations. The method is extended to analyses involving thermal body force loading and initial stress loading (e.g. plasticity). Therefore, in a thermal or nonlinear problem, the user has the option to use either conventional volume integration or the new particular integral approach for incorporating the body force effect into the analysis.

The governing differential equation for a homogenous, isotropic body subjected to body force loading can be expressed in operator notation as [35,38,44,45,52]:

$$L(u_i) + f_i = 0 \tag{2.1.5}$$

where

$L(u_i) = (\lambda + \mu)u_{i,ji} + \mu u_{i,jj}$

u_i represents the displacement vector,

$(f_1, f_2, f_3) = \rho\omega^2(x_1, x_2, 0)$ for centrifugal loading about the z axis

$f_i = -\beta T_{,i}$ for thermoelastic loading, with $\beta = (3x + 2\mu)\alpha$ and

$f_i = -\sigma^o_{ij,j}$ for initial stress (nonlinear) problems

When more than one type of body force is present, the net body force is the summation of the individual body forces.

A solution satisfying a linear, inhomogenous, differential equation and boundary conditions can be obtained using the method of particular integrals if the complete solution of the corresponding homogenous equation is known, provided of course, a particular solution can be found. Therefore, the solution of the above equation can be represented as the sum of a complementary function u_i^c satisfying the homogenous equation [35,38,44,45,46]

$$L(u_i^c) = 0 \tag{2.1.6}$$

and a particular integral u_i^p satisfying the inhomogenous equation

$$L(u_i^p) = f_i \tag{2.1.7}$$

The total solution u_i is expressed as

$$u_i = u_i^c + u_i^p \tag{2.1.8}$$

The boundary integral equation satisfying the homogenous part of the differential equation is the complementary function in this procedure. The complementary function for displacement at point ξ, is expressed as

$$C_{ij}(\xi)u_i^c(\xi) = \int_s [G_{ij}(x,\xi)t_i^c(x) - F_{ij}(x,\xi)u_i^c(x)]dS(x)$$

$$i,j = 1,2,3 \tag{2.1.9}$$

where the u_i^c and t_i^c are the complementary functions for displacement and traction, respectively. The total solution for displacement u_i and traction t_i are

$$u_i = u_i^c + u_i^p$$

$$t_i = t_i^c + t_i^p \tag{2.1.10}$$

where u_i^p and t_i^p are the particular integrals for displacement and traction, respectively. The particular integral is classically found via the method of undetermined coefficients, the method of variation of parameters, or obtained by inspection of the inhomogenous differential equation. In the theory of linear, inhomogenous, differential equations, it is understood that the particular integral is not unique, and any expression satisfying equation (2.1.5) is a particular integral, regardless of boundary conditions or how it was obtained.

Particular integrals for centrifugal loading are relatively simple to derive and are given in many text books on elasticity. On the other hand, particular integrals satisfying the inhomogenous differential equation for thermal and initial stress loading are more complex since the distribution of temperature and initial stress are general in nature. To accommodate this general distribution, a device known as the global shape function is employed. It is then possible to derive the particular integrals for displacement, stress, strain and traction in terms of the inhomogenous quantity (either temperature or initial stress).

Once a particular integral is found it is added to the complementary function to form the total solution. The parameters of the complementary function are adjusted to insure that the total solution satisfies the boundary condition, and hence, produces a unique solution to the boundary value problem.

The boundary integral equation (2.1.9) is discretized and integrated for a system of boundary nodes in the manner described in Chapter 3. The resulting equations can be expressed in matrix form as

$$Gt^c - Fu^c = 0 \tag{2.1.11}$$

By introducing equation (2.1.10) into equation (2.1.11), the complementary function can be eliminated;

$$Gt - Fu = Gt^p - Fu^p \tag{2.1.12}$$

The particular integral terms on the right hand side of this equation are functions of known body forces.

In a multi-region problem a set of (complementary) equations, similar to equation (2.1.11), are generated independently for each region. Likewise, the particular integrals of each region are derived independently leading to a set of equations for each region similar in form to equation (2.1.12).

The particular integral approach is also utilized in **BEST3D** to represent hotspots and discontinuities within the volume. These are taken care of by developing the necessary particular integrals due to an initial stress system σ_{ij}^o within a rectilinear prism of dimension $a \times b \times c$. The solution can be expressed as [24]:

$$\begin{aligned} u_i(\xi) &= H_i(x,\xi)T(x) \\ \sigma_{ij}(\xi) &= H_{ij}^\sigma(x,\xi)T(x) \end{aligned} \tag{2.1.13}$$

where T is the temperature of the hotspot over a volume $a \times b \times c$ and

$$\begin{aligned} u_i(\xi) &= K_{ijk}(x,\xi)\sigma_{jk}^o(x) \\ \sigma_{ij}(\xi) &= K_{ijkl}^\sigma(x,\xi)\sigma_{kl}^o(x) \end{aligned} \tag{2.1.14}$$

where $\sigma_{jk}^o(x)$ are determined by satisfying the conditions of zero stress within the discontinuity. The functions H, H^σ, K and K^σ are continuous functions of x and ξ and no additional integrations are involved [24].

Elastic analysis of three-dimensional bodies can also be carried out in the presence of embedded holes and inserts as described later in this chapter.

2.2 INELASTIC ANALYSIS

For structural problems which involve inelastic material response, volume integrals must be retained over at least a portion of the body. In these cases, equations (2.1.3) and (2.1.4) are rewritten, in incremental form, in terms of the initial stresses as [18-25]:

$$C_{ij}\dot{u}_i(\xi) = \int_S [G_{ij}(x,\xi)\dot{t}_i(x) - F_{ij}(x,\xi)\dot{u}_i(x)]ds + \int_V B_{ijk}(x,\xi)\dot{\sigma}^o_{ik}(x)dv \tag{2.2.1}$$

and

$$\dot{\sigma}_{ij}(\xi) = \int_S [G^\sigma_{kij}(x,\xi)\dot{t}_k(x) - F^\sigma_{kij}(x,\xi)\dot{u}_k(x)]ds + \int_{V-V_\epsilon} M_{ijkl}(x,\xi)\dot{\sigma}^o_{kl}(x)dv + J_{ijkl}\dot{\sigma}^o_{kl}(\xi) \tag{2.2.2}$$

where $\dot{\sigma}^o_{kl}$ is a jump term arising out of analytical treatment of the volume integral over a small spherical region around ξ [25].

The volume integrals in (2.2.1) and (2.2.2) both involve unknown initial stresses, whose evaluation involve the use of the nonlinear constitutive model. Thus equations (2.2.1), (2.2.2) and the nonlinear constitutive model can be considered as a single nonlinear system to be solved for the unknown boundary displacements and tractions and for the initial stresses within the inelastic region.

In **BEST3D** the material models provided are:

(a) Nonlinear strain hardening Von Mises model [2,7,16]
(b) Multi-surface cyclic plasticity model [36]
(c) Thermally sensitive viscoplastic model of Walker [49]

The pertinent details for these models arc provided below:

(a) <u>Von Mises Model</u>

This model was developed from observations on metal behavior and assumes that yielding begins when the distortional energy reaches the distortional energy at yield in simple tension. The yield criterion is given by

$$F(J_2) = \frac{1}{2}S_{ij}S_{ij} - \frac{1}{3}\sigma_o^2 = 0 \tag{2.2.3}$$

where

$S_{ij} = \sigma_{ij} - \frac{1}{3}\delta_{ij}\sigma_{kk}$

σ_o = uniaxial yield strength in simple tension.

Plastic flow is defined through a flow rule which relates the plastic strain rate tensor to the gradient of a plastic potential through the stress point. In metals, this potential

is considered considered identical to the yield function and the associated plastic strain rate becomes normal to the yield surface at a stress point. This normality condition is expressed as

$$\dot{\varepsilon}_{ij} = \dot{\lambda}\frac{\partial F}{\partial \sigma_{ij}} \tag{2.2.4}$$

where $\dot{\lambda}$ is a non-negative scalar variable which depends upon the current stress state and the past history of loading. An isotropic, strain hardening rule is utilized to describe the subsequent yield surfaces resulting from continuous plastic distortion.

The isotropic hardening theory assumes that the yield surface uniformly expands about the origin in stress space while maintaining its shape, center and orientation, during plastic flow.

For an isotropic hardening material, the yield function can be written as

$$F(\sigma_{ij}, \varepsilon^{p}_{ij}, h) = 0 \tag{2.2.5}$$

where h is the hardening parameter that represents the loading history in the plastic range.

Utilizing the above stated concepts along with a consistency condition that every subsequent stress point beyond the yield stress point must lie on an expanded yield surface, the increment of stress for a given mechanical strain increment is obtained as

$$\dot{\sigma}_{ij} = D^{ep}_{ijkl}\dot{\varepsilon}^{m}_{kl} \tag{2.2.6a}$$

where

$$\dot{\varepsilon}^{m}_{kl} = \dot{\varepsilon}^{elastic}_{kl} + \dot{\varepsilon}^{plastic}_{kl} \tag{2.2.6b}$$

and D^{ep}_{ijkl} is the plastic modulus tensor which can be explicitly written as

$$D^{ep}_{ijkl} = 2\mu[\delta_{ik}\delta_{jl} + \frac{\nu}{1-2\nu}\delta_{ij}\delta_{kl} - \frac{3S_{ij}S_{kl}}{2\sigma_o^2(1+h/3\mu)}] \tag{2.2.7}$$

(b) Two-surface Cyclic Plasticity Model

The standard isotropic and kinematic hardening models, which are based on distinctively defined elastic and elastoplastic behavior, are inadequate to represent the smooth transition from elastic to fully plastic state that is observed experimentally during cyclic loading of most metals and geologic materials. As a result many other models which are conceptually similar to the nested yield surface model originally proposed by Mroz [46] involving both kinematic and isotropic hardening have been developed. An extremely practical model based on the ideas of Mroz [46] was introduced by Kreig [47] and Dafalias and Popov [48] which they described as the two surface model. The model used here is essentially a modified version [36] of that described by Kreig and is briefly described below.

In this model it is assumed that there exists a loading or yield surface in the domain bounded by another limiting yield or bounding surface. During loading in the plastic state the stress state remains in contact with the inner yield surface, which translates in the stress space during deformation. Once this surface touches the bounding surface, the behaviour during deformation is governed by either isotropic or kinematic hardening rules. Thus, during deformation, the stress point moves through three distinct regions: (a) an elastic region, which is associated with recoverable strains, when the stress state is within the inner yield surface; (b) a metaelastic region, where the hardening is determined from the relative location of the stress state in relation to the bounding surface, when the stress point is on the inner yield surface, but within the bounding surface; and (c) a plastic region, where the stiffness is given by plastic hardening parameter associated with the bounding surface when the stress point is on the bounding surface. The stiffness of the material changes dramatically as the stress state moves through the regions.

In the elastic region, the behavior is governed by elastic constitutive relationships which are well known. The rules governing the behavior in the metaelastic region, which provide the smooth transition from elastic to plastic behavior, is determined as follows.

The yield functions associated with the loading and bounding surfaces can be expressed, respectively (both having the same shape and orientation) as:

$$F(\bar{\sigma}_{ij}, \epsilon^p_{ij}) = 0 \tag{2.2.8a}$$

and

$$F(\sigma^b_{ij}, \epsilon^p_{ij}) = 0, \tag{2.2.8b}$$

where

$\bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$,

σ_{ij} = stress state on the loading surface,

σ^b_{ij} = stress state on the bounding surface,

ϵ^p_{ij} = plastic strain,

α_{ij} = location of the center of loading (inner yield) surface.

During deformation from the stress state A, the translation of the loading surface is along AA^b (see Fig. 2.2.1), where A^b is a unique image of the current stress state on the bounding surface. The location of A^b is determined by the intersection of a line drawn parallel to OA through O′ and the bounding surface. The translation of the loading surface is then given by [36]:

$$\dot{\alpha}_{ij} = \frac{\dot{\mu}_o}{\sigma_o}[(\sigma^b_o - \sigma_o)\sigma_{ij} - (\alpha_{ij}\sigma^b_o - \alpha^b_{ij}\sigma_o)], \tag{2.2.9}$$

where

σ_o = an equivalent yield stress associated with the loading surface,

σ_o^b = an equivalent yield stress associated with the bounding surface,
α_{ij} = location of the center of loading surface,
α_{ij}^b = location of the center of bounding surface,
$\dot{\mu}_o$ = a scalar parameter yet to be determined.

Since the stress point remains on the loading surface during deformation, the consistency conditions must be satisfied. This gives the value of:

$$\dot{\mu}_o = \frac{(\partial F/\partial \sigma_{ij})\dot{\sigma}_{ij}}{\frac{\partial F}{\partial \sigma_{kl}}(\sigma_{kl}^b - \sigma_{kl})} \tag{2.2.10}$$

Using the flow rule and consistency condition, the governing stress-strain relationship can be obtained as

$$\dot{\sigma}_{ij} = \left[D_{ijkl}^e - \frac{D_{ijmn}^e(\frac{\partial F}{\partial \sigma_{mn}^b})(\frac{\partial F}{\partial \sigma_{pq}^b})D_{pqkl}^e}{H_e + H_p} \right] \dot{\epsilon}_{kl}, \tag{2.2.11}$$

where

$$H_e = (\frac{\partial F}{\partial \sigma_{ij}^b}) D_{ijkl}^e (\frac{\partial F}{\partial \sigma_{kl}^b}), \tag{2.2.12}$$

and H_p is the plastic hardening modulus.

H_p is determined from the position of the stress state relative to the loading and bounding surfaces and the one-dimensional stress-strain behavior of the material as

$$H_p = h \left[\frac{\bar{\beta}}{\sigma_{ref}} \right]^n, \tag{2.2.13}$$

where h is the hardening of the bounding surface. β_{ij} represents the distance between the current stress state to the last location of the loading surface center when the material was elastic and $\bar{\beta} = [\beta_{ij}\beta_{ij}]^{1/2}$. σ_{ref} is the distance from the last location of the loading surface center for elastic behavior to the point A^b on the bounding surface (Fig. 2.2.1) obtained by drawing a line parallel to $\dot{\sigma}_{ij}$ and locating the intersection with the bounding surface.

The exponential power n is given by the expression

$$n = \frac{log(inner\ surface\ hardening\ /\ outer\ hardening)}{log(inner\ yield\ stress\ /\ outer\ yield\ stress)} \tag{2.2.14}$$

During cyclic loading, the loading and unloading are determined from the sign (positive for loading and negative for unloading) of the loading function defined as

$$dL = N_{ij}\dot{\sigma}_{ij} \tag{2.2.15}$$

where N_{ij} is the normal at σ_{ij}^b and is given by

$$N_{ij} = \frac{\partial F / \partial \sigma_{ij}^{b}}{\left[\frac{\partial F}{\partial \sigma_{kl}} \frac{\partial F}{\partial \sigma_{kl}}\right]^{1/2}} \qquad (2.2.16)$$

Equations (2.2.9)-(2.2.16) provide the necessary and complete mechanical description of the material for monotonic as well as cyclic loading and requires only three inelastic parameters h, n and σ_{ref}.

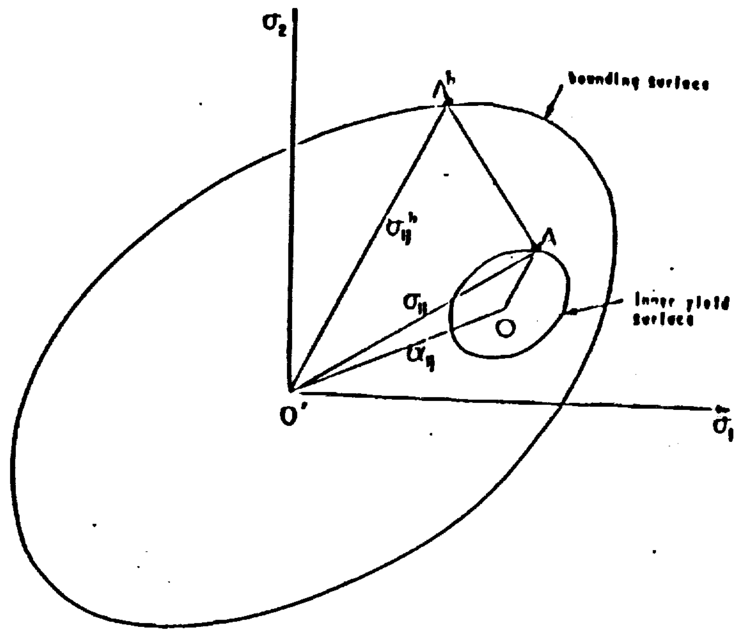


Fig. 2.2.1 Two surface plasticity model

(c) <u>The Walker Viscoplastic Model</u>

The thermally sensitive Walker viscoplasticity model which is implemented in **BEST3D** is described in detail in Walker[49] and Cassenti and Thompson [50].

Reference [49] describes the basic theory; while Reference [50] describes modifications to the form of the basic theory, and modifications to the material parameters for Hastelloy X. The modifications provide more accuracy at relatively low temperatures.

The inelastic strain consists of two components: a time dependent power law creep component, containing the material constants n and k, and a time independent plastic component. The back stress Ω is a key variable in many viscoplastic material models. Its evolution is given below:

$$\dot{\epsilon}^p_{ij} = \frac{(3/2s_{ij} - \Omega_{ij})}{\sqrt{2/3(3/2s_{ij} - \Omega_{ij})(3/2s_{ij} - \Omega_{ij})}} \quad \frac{\left\{exp\left[\frac{\sqrt{2/3(3/2s_{ij} - \Omega_{ij})(3/2s_{ij} - \Omega_{ij})}}{K}\right] - 1\right\}}{\beta}$$

$$K = K_1 - K_2 e^{-n_7 R} - K_3 e^{-n_8 L}$$

$$\dot{L} = \left|\sqrt{2/3\dot{\epsilon}^p_{ij}\dot{\epsilon}^p_{ij}} - \left|\sqrt{2/3\dot{\epsilon}^p_{ij}\dot{\epsilon}^p_{ij}}\right|\right|$$

$$\dot{R} = \sqrt{2/3\dot{\epsilon}^p_{ij}\dot{\epsilon}^p_{ij}}$$

$$\Omega_{ij} = \Omega_{ij0} + \Omega_{ij1} + \Omega_{ij2}$$

$$\begin{aligned}\dot{\Omega}_{ij1} = n_2\dot{\epsilon}^p_{ij} - \Omega_{ij1}\Big(\Big[n_3 + n_4 e^{-n_5|log\dot{R}/\dot{R}_o|}\Big] \\ \dot{R} + n_6 + \frac{1}{n_2}\frac{\partial n_2}{\partial \Theta}\dot{\Theta} - \frac{1}{n_3}\frac{\partial n_3}{\partial \Theta}\dot{\Theta}\Big) \\ + \left(\frac{n_2}{n_3} - \Omega_{ij1}\right)\frac{\partial n_3}{\partial \Theta}\epsilon^p_{ij}\dot{\Theta}\end{aligned}$$

$$\begin{aligned}\dot{\Omega}_{ij2} = n_{11}\dot{\epsilon}^p_{ij} - \dot{\Omega}_{ij2}\Big(n_9\dot{R} + n_{10} + \frac{1}{n_{11}}\frac{\partial n_{11}}{\partial \Theta}\dot{\Theta} - \frac{1}{n_9}\frac{\partial n_9}{\partial \Theta}\dot{\Theta}\Big) \\ + \left(\frac{n_{11}}{n_9} - \Theta_{ij2}\right)\frac{\partial n_9}{\partial \Theta}\epsilon^p_{ij}\dot{\Theta}\end{aligned}$$

$$\begin{aligned}\dot{\Omega}_{ij} = 3\dot{\Omega}\Big[\frac{\dot{C}_{ik}C_{kj}}{C_{pq}C_{pq}} + \frac{C_{ik}\dot{C}_{kj}}{C_{pq}C_{pq}} - \left(\frac{2c_{ik}C_{kj}}{C_{pq}C_{pq}}\right)\left(\frac{C_{rs}\dot{C}_{rs}}{C_{uv}C_{uv}}\right)\Big] \\ + \Big[3\frac{C_{ik}C_{kj}}{C_{pq}C_{pq}} - \delta_{ij}\Big]\frac{\partial \dot{\Omega}}{\partial \Theta}\dot{\Theta}\end{aligned}$$

$$s_{ij} = \sigma_{ij} - \frac{1}{3}\delta_{ij}\sigma_{kk}$$

Material Constants: $\lambda, \mu, \dot{\Omega}, \beta, n_1, n_2, n_3, n_4, n_5, n_6, n_7,$
$n_8, n_9, n_{10}, n_{11}, K_1, K_2,$ *and* K_3 depend on temperature, Θ.

Inelastic Solution Algorithms:

Two inelastic solution algorithms have been implemented in **BEST3D** to solve the nonlinear system. The first method is an iterative procedure which includes a time saving feature that reduces the number of iterations needed for convergence, by utilizing the past history of initial stresses to estimate the value of initial stress rates for the following load increment. This method is applicable to all of the material models listed above.

The second procedure is a direct, variable stiffness algorithm that does not require iteration. Instead, the procedure exploits certain features of the constitutive relationship to express the unknown nonlinear initial stress tensor as a scalar quantity which can then be eliminated from the boundary equation system through a back substitution of the (modified) stress rate equation. This method is restricted to the Von Mises model (with strain hardening) under monotonic loading conditions. Explicit details can be found in references [25,37,39,40,45]

2.3 FREE-VIBRATION ANALYSIS

A new method for free-vibration analysis by boundary element method using particular integrals has been implemented in the current version of **BEST3D** . This method utilizes a fictitious vector function to approximate the inertia forces and then uses the well known concept of complementary functions and particular integrals to solve the resulting governing differential equations [29,53].

The governing differential equation for free-vibration of an elastic, homogenous and isotropic solid can be written in operator notation as

$$L(u_i) + \rho w^2 u_i = 0 \tag{2.3.1}$$

The solution of the above equation can be represented as the sum of a complementary function, u_i^c, satisfying

$$L(u_i^c) = 0 \tag{2.3.2}$$

and a particular integral, u_i^p, satisfying

$$L(u_i^p) + \rho w^2 u_i = 0 \tag{2.3.3}$$

In equation (2.3.3), the unknown displacement u_i within the domain is approximated by an infinite summation of the products of an unknown fictitious density function, ϕ_k, and a known function D_{ik}. More specifically

$$u_i(x) \simeq \sum_{m=1}^{\infty} D_{ik}(x, \xi^m)\phi_k(\xi^m) \tag{2.3.4}$$

The direct boundary integral equation related to the displacement function u^c can be written as

$$C_{ij}u^c(\xi) = \int_S [G_{ij}(x,\xi)t_i^c(x) - F_{ij}(x,\xi)u^c(x)]ds \tag{2.3.5}$$

where G_{ij} and F_{ij} are the elasto-static kernels (i.e. fundamental solutions of eq. 2.3.2).

By usual discretization of the boundary S, equation (2.3.6) can be expressed in a matrix form as

$$[G]\{t^c\} - [F]\{u^c\} = \{0\} \tag{2.3.6}$$

The complementary solutions are related to the real displacement u_i and tractions t_i via

$$u_i^c = u_i - u_i^p \tag{2.3.7}$$

$$t_i^c = t_i - t_i^p \tag{2.3.8}$$

Substituting equations (2.3.4), (2.3.7) and (2.3.8) into equation (2.3.6), the following equations [29] are obtained

$$[G]\{t\} - [F]\{u\} = \rho w^2[M]\{u\} \tag{2.3.9}$$

By incorporating the known boundary conditions (i.e. at a node either $t_i = 0$ or $u_i = 0$) in eq. 2.3.9, an algebraic expression results for eigenvalue extraction, i.e.

$$[A]\{x\} = \rho w^2\boxed{M}\{x\} \tag{2.3.10}$$

Finally, equation (2.3.10) is solved by using an eigenvalue extraction routine developed by Garbow of Argonne National Laboratory to obtain the eigen frequencies and mode shapes.

2.4 PERIODIC DYNAMIC ANALYSIS

By using the dynamic reciprocal theorem, the boundary integral representation for periodic dynamics can be written in the frequency domain as Banerjee and Butterfield [2]:

$$C_{ij}u_i(\xi,\omega) = \int_S [G_{ij}(x,\xi,\omega)\ t_i(x,\omega) - F_{ij}(x,\xi,\omega)\ u_i(x,\omega)]\ dS(x) \tag{2.4.1}$$

where S is the surface enveloping the body and the fundamental solution G_{ij} and F_{ij} are defined in Banerjee et al [41] and Ahmad and Banerjee [51]. It should be noted here that although the functions G_{ij} and F_{ij} becomes identical to their static counterpart as ω tends to zero, it is important to evaluate this limit carefully because of the presence of ω in the denominator.

Once the boundary solution is obtained, the interior version of equation (2.4.1) can also be used to find the interior displacements; and the interior stresses can then be obtained using the strain displacement and stress strain relations as:

$$\sigma_{jk}(\xi,\omega) = \int_S [G^{\sigma}_{ijk}(x,\xi,\omega)\ t_i(x,\omega) - F^{\sigma}_{ijk}(x,\xi,\omega)\ u_i(x,\omega)]\ dS(x) \tag{2.4.2}$$

The stresses at the surface can be calculated by combining the constitutive equations, the directional derivatives of the displacement vector and the values of field variables in an accurate matrix formulation.

The boundary integral formulations (2.4.1) and (2.4.2) can also take account of internal viscous dissipation of energy (damping) using complex elastic moduli in the usual manner.

2.5 TRANSIENT DYNAMIC ANALYSIS

The direct boundary integral formulation for a general, transient, elastodynamic problem can be constructed by combining the fundamental point-force solution (Stokes solution) of the governing equation of motion (Navier-Cauchy equations) with the dynamic reciprocal theorem. For zero initial conditions and zero body forces, the boundary integral formulation for transient elastodynamics reduces to [2,26,27,28]

$$C_{ij}u_i(\xi,T) = \int_S [G_{ij}(x,\xi,T) * t_i(x,T) - F_{ij}(x,\xi,T) * u_i(x,T)] \; dS(x) \tag{2.5.1}$$

where

$$G_{ij} * t_i = \int_0^T G_{ij}(x,T;\xi,\tau) \; t_i(x,\tau) \; d\tau \tag{2.5.2a}$$

$$F_{ij} * u_i = \int_0^T F_{ij}(x,T;\xi,\tau) \; u_i(x,\tau) \; d\tau \tag{2.5.2b}$$

are Riemann convolution integrals, while ξ and x are the space positions of the receiver (field point) and the source (source point). The fundamental solutions G_{ij} and F_{ij} are the displacements u_i and tractions t_i at a point x at time T due to a unit force vector applied at a point ξ at a preceding time τ. The tensor C_{ij} is the discontinuity (or jump) tensor, arising from the singularity of the F_{ij} kernel.

Equation (2.5.1) represents an exact formulation involving integration over the surface as well as the time history. It should also be noted that this is an implicit time-domain formulation because the response at time T is calculated by taking into account the history of surface tractions and displacements up to and including the time T. Furthermore, equation (2.5.1) is valid for both regular and unbounded domains.

This formulation has two major advantages:

1 - It is expressed entirely in terms of real variables, allowing much improved computational efficiency.

2 - Causality is preserved:, that is, the structure of the mathematical formulation is such that events at a given time can be affected only by prior history.

Once the boundary solution is obtained, the stresses at the boundary can be calculated by combining the constituitive equations, the directional derivatives of the displacement vector and the values of field variables in an accurate matrix formulation. For calculating displacements at interior points equation (2.5.1) can be used with $C_{ij} = \delta_{ij}$, and the interior stresses can be obtained from

$$\sigma_{jk}(\xi,T) = \int_S [G^{\sigma}_{ijk}(x,\xi,T) * t_i(x,T) - F^{\sigma}_{ijk}(x,\xi,T) * u_i(x,T)] \; dS(x) \tag{2.5.3}$$

In order to obtain the transient response at a time T_N, the time axis is discretized into N equal time intervals, i.e.

$$T_N = \sum_{n=1}^{N} n\Delta T \tag{2.5.4}$$

where ΔT is the time step.

Utilizing equations (2.5.4), and (2.5.2), equation (2.5.1) can be written as

$$C_{ij}u_i(\xi, T_N) - \int_{T_{N-1}}^{T_N} \int_S [G_{ij}t_i - F_{ij}u_i] \ dSd\tau = \int_{\tau=0}^{T_{N-1}} \int_S [G_{ij}t_i - F_{ij}u_i] \ dSd\tau \tag{2.5.5}$$

where the integral on the right hand side is the contribution due to the past dynamic history.

It is of interest that equation (2.5.5), like equation (2.5.1) still remains an exact formulation of the problem since no approximation has yet been introduced. However in order to solve equation (2.5.5) one has to approximate the time variation of the field quantities in addition to the usual approximation of spatial variation. For this purpose a linear interpolation function is used. The details can be found in the paper by Ahmad and Banerjee [28].

2.6 HEAT CONDUCTION ANALYSIS

The governing differential equation for transient heat conduction can be written as

$$k\frac{\partial^2 T}{\partial x_i \partial x_i} - \rho c \frac{\partial T}{\partial t} + \psi = 0, \tag{2.6.1}$$

with temperature T, conductivity k, density ρ, specific heat c, and body sources ψ. The corresponding reciprocal theorem

$$\int_S q(x) * T^*(x)ds + \int_V \psi(x) * T^*(x)dv = \int_S q^*(x) * T(x)ds + \int_V \psi^*(x) * T(x)dv \tag{2.6.2}$$

relates any two states of temperature T, heat flux q, and ψ body sources occupying in the volume V with surface S. Note that since convolutions appear in (2.6.2), the entire time history is involved. Using the infinite space fundamental solution of (2.6.1) as one of the states, and the solution of the desired boundary value problem with zero body sources as the other, produces the following boundary integral equation:

$$C(\xi)T(\xi,t) = \int_S [G(x,\xi,t) * q(x,t) - F(x,\xi,t) * T(x,t)]ds(x). \tag{2.6.3}$$

In (2.6.3), the kernel functions G and F are derived from the fundamental solutions, while $C(\xi)$ is the discontinuity term arising from the singularity of the F kernel.

Under steady-state conditions the governing equation simplifies to

$$k\frac{\partial^2 T}{\partial x_i \partial x_i} + \psi = 0, \tag{2.6.4}$$

and the boundary integral equation becomes

$$C(\xi)T(\xi) = \int_S [G(x,\xi)q(x) - F(x,\xi)T(x)]ds(x). \tag{2.6.5}$$

G and F are now the well-known potential flow kernel functions. Equation (2.6.5) is, of course, time independent. For more information, please refer to Banerjee and Butterfield [2] and Dargush and Banerjee [42].

BEST3D contains multiregion capability for both steady-state and transient heat conduction as boundary-only problems. The spatial and temporal discretization of (2.6.5) and (2.6.3) is discussed in some detail in Chapter 3.

The application of the boundary element method for heat conduction analysis effectively reduces the dimensionality of the problem by one. However, in certain instances, further reduction is possible. For example, in axisymmetric problems, the required circumferential integration can be performed in a semi-analytical manner at the kernel level. As a

result, numerical integration is only needed along the boundary in the r-z plane. Similarly, in other situations, meaningful approximations can be introduced to simplify an analysis. Such is the case for circular, small diameter holes embedded in a three-dimensional heat conducting body.

Consider a cylindrical exclusion in an infinite region. The boundary integral equation (2.6.3) is valid at any point ξ, where S represents the surface of the exclusion. Remote from the hole, it is reasonable to assume constant temperature and flux in the circumferential direction along the surface of the hole. With that assumption, the integration required by (2.6.3) in the circumferential direction can be performed in a semi- analytical manner and incorporated in the kernel functions. Ignoring the contribution of the circular disks at the ends of the hole, all that remains is a line integral along the centerline C. Thus,

$$C(\xi)T(\xi,t) = \int_G [G^c(x,\xi,t) * q(x,t) - F^c(x,\xi,t) * T(x,t)]dc(x), \tag{2.6.6}$$

where the kernels G^c and F^c include the circumferential integration. The remaining line integration can be performed numerically. In general, there can be multiple exclusions within a finite three- dimensional body. The boundary integral equation for a region with outer boundary S and N holes generalizes to

$$\begin{aligned} C(\xi)T(\xi,t) = & \int_S [G(x,\xi,t) * q(x,t) - F(x,\xi,t) * T(x,t)]ds(x) \\ & + \sum_{n=1}^{n} \left[G^c(x,\xi,t) * q(x,t), -F^c(x,\xi,t) * T(x,t) \right] dc_n(x). \end{aligned} \tag{2.6.7}$$

The embedded hole capability is available for both steady-state and transient problems.

It should be noted that many other physical processes are governed by equations of the form of (2.6.1) or (2.6.4). For example, the steady flow of inviscid incompressible fluids reduces to (2.6.4), as does problems of electrostatics, while the diffusion of chemical substance is represented by (2.6.1). In all of these cases, with an appropriate translation of the field variables, **BEST3D** is directly applicable.

2.7 STRESS ANALYSIS WITH EMBEDDED HOLES

The regular boundary integral equation for displacement of a three- dimensional body with embedded holes may be written as

$$C_{ij}u_i(\xi) = \int_S \Big[G_{ij}(x,\xi)t_j(x) - F_{ij}(x,\xi)u_j(x)\Big]dS(x) + \sum_{m=1}^{M}\int_{S^m} G_{ij}(x,\xi)t_j(x)dS^m(x) - \sum_{m=1}^{M} F_{ij}(x,\xi)u_j(x)ds^m(x) \tag{2.7.1}$$

where G_{ij} and F_{ij} is the Kelvin point force solution and corresponding traction kernel

u_i and t_i are boundary displacement and tractions

S is the outer surface of the body

s_m is the surface of the m^{th} hole

C_{ij} are the constants determined by the relative smoothness of the body

If body forces or nonlinear effects are present they can be incorporated in equation (2.7.1) through a volume integral or using a particular integral.

Discretization of equation (2.7.1) in the usual boundary element manner requires a very fine discretization of the holes. Alternatively, **BEST3D** offers a new concept called 'Hole Elements' for the treatment of the embedded holes. This formulation is derived from equation (2.7.1) employing a circular shape function and appropriate transformations and analytical integration about the circumference of the hole. In this formulation the following assumptions are made. First, the hole is of a tubular shape and is assumed to be of circular cross section and closed at the ends with a flat circular surface. The displacement about the circumference of the hole is assumed to be of a sine-cosine variation and the pressure is assumed to be constant. The variation of the geometry and field variables in the longitudinal direction is linear or quadratic.

A hole which has curvature along its length will differ in surface area about the circumference on the curved portion of the tube. This is neglected in the integration since the integration is performed on an axisymmetric ring in which the surface area is constant about the circumference. This error, however, is small and disappears completely on a straight tubular hole which is most commonly encountered. Finally, we note that the holes should not intersect the outer surface of the body or intersect other holes. This minor restriction can be lifted if results at these locations are not of interest. As long as the holes do not coincide with nodal points of other elements, the errors will be localized and will not effect the overall boundary element solution.

The essential part of the formulation is the conversion of the two- dimensional surface integration of the hole to a one-dimensional integration. In equation (2.7.1) the last two

integrals under the summation are the integrals associated with the hole which are to be modified. In the present formulation if the holes in the three-dimensional body are assumed traction free then only the F_{ij} (traction) integral needs to be derived. If pressure is present in the holes then the G_{ij} (displacement) kernel is also necessary.

The kernel functions for the holes are derived by analytically integrating the three-dimensional kernels in the circumferential direction. To facilitate the integration the kernel functions are first expressed in local coordinates with the center of the coordinate system coinciding with the center of the hole and the z axis aligned with the centerline of the tube. The relative translation ξ_i' is added to the field coordinate ξ_i and the rotation is applied using the appropriate vector transformation.

$$\xi_i = a_{ij}\bar{\xi}_j + \xi_i'$$

where a_{ij} are the direction cosines between the axis of the local and global coordinate systems and the bar indicates a local variable.

The displacement must also be transformed between the local and the global systems by

$$u_i = a_{ik}\bar{u}_k$$

or

$$\bar{u}_j = a_{mj}u_m$$

the integration point x_i for a ring can now be expressed in cylindrical coordinates relative to the center of the hole as

$$X_1 = Rcos\theta$$

$$X_2 = Rsin\theta$$

$$X_3 = z$$

where R represents the radius of the hole, i.e., $R = (x_1^2 + x_2^2)^{1/2}$.

The normal vectors are transformed by

$$n_1 = n_r cos\theta$$

$$n_2 = n_r sin\theta$$

$$n_3 = n_z$$

where n_r and n_z represents the normals of the side of the hole in local coordinates and are dependent on the change in the radius of the hole. For a straight hole $n_r = 1$ and $n_z = 0$, and on the flat surface closing the end of the hole $n_r = 0$ and $n_z = 1$. Next a circular shape function is employed to approximate the variation in the displacement about the circumference of the hole. The circular shape function which is multiplied and integrated

with the 3-D F_{ij} kernel, allowing the nodal values to be brought outside the integral. The shape function is expressed as

$$u_i = M^\gamma U_i^\gamma (\text{ summation is implied }, \gamma = 1,2,3)$$

where

$$M^1(\theta) = \frac{1}{3} + \frac{2}{3} cos\theta$$

$$M^2(\theta) = \frac{1}{3} + \frac{\sqrt{3}}{3} sin\theta - \frac{1}{3} cos\theta$$

$$M^3(\theta) = \frac{1}{3} - \frac{\sqrt{3}}{3} sin\theta - \frac{1}{3} cos\theta$$

and U_i^γ are the nodal displacements.

If internal pressure is present in the hole, the pressure is assumed to be constant about the circumference. The traction vector in equation (2.7.1) however, is written as a global vector and must be converted to a local scalar variable. This transformation vector is multiplied and integrated along with the G_{ij} kernel allowing the constant pressure value to be brought outside the integration. The load transformation is

$$t_i = N_i P$$

where N_i is the transformation vector of the normal vectors.

$$N_1 = n_r cos\theta$$

$$N_2 = n_r sin\theta$$

$$N_3 = n_z$$

and P is the internal pressure in the hole. If the pressure in the hole is zero, the second last term under the summation in equation (2.7.1) vanishes.

The last two terms in equation (2.7.1) can now be analytically integrated in the cirumferential direction. For the m^{th} hole this can be expressed as

$$\begin{aligned}\int_{S^m} G_{ij}(x,\xi)t_j(x)ds^m(x) &= \int_{C^m} a_{ik} \int_0^{2\pi} G_{kj}(R,\theta,z,\xi)N_j R d\theta p dC^m \\ &= \int_{C^m} G_i^H(R,z,\xi) p dC^m(z)\end{aligned}$$

$$\begin{aligned}\int_{S^m} F_{ij}(x,\xi)u_j(x)ds^m(x) &= \int_{C^m} a_{ik} \int_0^{2\pi} F_{kj}(R,\theta,z,\xi)M^\gamma R d\theta a_{mj} U_m^\gamma dC^m \\ &= \int_{C^m} F_{ij}^{H\gamma}(R,z,\xi) U_j^\gamma dC^m(z)\end{aligned}$$

where C^m represents the curvilinear centerline of the hole and G_{ij}^H and F_{ij}^H represent the analytically integrated hole kernels. Note, since the transformation vector a_{ik} is independent of θ, it may be taken outside the $d\theta$ integration.

The analytic circumferential integration of the kernels yields functions containing elliptical integrals and related functions. The elliptical integrals and related functions are expressed numerically by common series approximations.

Additional series approximations had to be derived for a range of input values (coordinate locations) that were found to cause numerical instabilities in these common approximations. The new series were derived using a best fit polynomial approximation (as a function of elliptic moduli) using values of the integrals calculated by very accurate numerical integration in the circumferential direction.

The derivation of the hole kernels corresponding to stress or strain equation can be calculated in two ways. In the first method the strain equation is derived from the displacement equation (2.7.1) by differentiation and application of the strain-displacement equation. The stress equation is then found using Hooke's Law. However, due to the complexity of the kernel functions of the displacement equation, an alternative method is employed in which appropriate transformations and integration in the circumferential is carried out in the manner identical to the derivation of the displacement kernel functions for hole element.

In constructing a boundary element system, the equation (2.7.1) is usually written for all nodal points used in the representation of the boundary and hole field variables. Hence, for every node used along the length of a hole element, three (three degree of freedom) equations must be written since the circular shape function's variations three nodes. Therefore, for a hole, a single quadratic (3-nodal) shape function in the longitudinal direction would result in a total of nine (3 degree of freedom) equations.

The equations for the nodes on the outer boundary are written at the coordinates of the nodes in the usual BEM manner. The jump terms and the Cauchy-principle-value of the singular integration associated with the singular elements are evaluated using the rigid body translation technique. In the present formulation, the summation of the coefficients corresponding to the holes should be zero for the equations written for the nodes on the outer boundary. It is recommended to assume the summation to be exactly zero and not to include any residue from the summation of the coefficients of the holes in the rigid body calculation for the points on the outer boundary.

The equations written for the nodes defining the hole may be handled in a similar manner. However, small errors may be introduced into the calculation of the hole coefficients of the diagonal terms either due to errors in the singular integration of the complex hole kernels or due to the assumptions made approximating the integration of a curve tubular hole. In a rigid body coefficient calculation of the diagonal term of the **F** (traction) matrix, the summation of these errors may have some influence on the results. Therefore, it was decided to write the equations corresponding to the nodes of the holes slightly inside the hole (offset from the surface of the hole one-fourth the distance of the radius). Likewise,

equations for the nodes at the end of the hole are moved slightly off the surface of the end to inside the hole. Note, the nodal values of the displacement and pressures associated with the hole elements still lie on the surface of the hole. By moving the equation point inside the hole, the singular integration is circumvented and all the values of $C_{ij}(\xi)$ (equation (2.7.1)) are zero for these equations written inside the void of the hole. However, care is still required for the delicate integration of a hole element for a node contained in that element.

After all source nodes and the boundary element system have been integrated over all surface elements and hole elements (of a subregion) the system for all subregions may be assembled incorporating all special interface conditions (perfect body, springs, sliding, etc.) in the usual manner. All nodal displacement at the holes will be contained in the systems unknowns and any given boundary conditions of internal pressure within the hole will be multiplied by their respected coefficients and added to the right-hand side. Specified and unknown boundary conditions on the outer boundary are collected and assembled with their coefficients in the usual manner and the system may be solved yielding results for both the unknown conditions on the outer boundary and for the displacement on the surface of the holes.

Results for displacement, stresses and strains at points in the domain of the body may be calculated using the discretized integral equations. Results for points on the outer boundary or on the surface of the hole are best calculated using shape function and the boundary stress/strain calculation normally used in boundary element method.

2.8 STRESS ANALYSIS OF BODIES WITH COMPOSITE INSERTS

The conventional boundary integral equation for displacement is the starting point for the composite formulation. The displacement for a point ξ inside the elastic composite matrix is given below

$$C_{ij}(\xi)u_i(\xi) = \int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)dS(x)$$

$$+\sum_{n=1}^{N}\int_{S^n}[G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]dS^n(x) \qquad i,j=1,2,3 \tag{2.8.1}$$

where

G_{ij}, F_{ij} are the fundamental solutions of the governing differential equations of the ceramic matrix of infinite extent

C_{ij} are constants determined by the relative smoothness at ξ

u_i, t_i are displacements and tractions

S, S^n are surfaces of the matrix and holes (left for fiber) respectively

N is the number of fibers

The conventional boundary integral equation for displacement can also be written for each of the N insert fibers. For the displacement at a point ξ inside the m^{th} insert we can write

$$C_{ij}^m(\xi)\bar{u}_i(\xi) = \int_{S^m}[G_{ij}^m(x,\xi)\bar{t}_i(x) - F_{ij}^m(x,\xi)\bar{u}_i(x)]dS^m(x) \qquad i,j=1,2,3 \tag{2.8.2}$$

G_{ij}^m, F_{ij}^m are the fundamental solutions of the m^{th} insert

C_{ij}^m are constants determined by the relative smoothness at ξ in insert m

$\bar{u}_i, \bar{t}_i$ are displacement and tractions associated with the m^{th} insert

S^m the surface of the m^{th} insert

We next examine the interface conditions between the composite matrix and the insert. For a perfect bond the displacement of the matrix and the displacement of the inserts along the interface are equal and the tractions are equal and opposite.

$$\bar{u}_j(x) = u_j(x) \tag{2.8.3a}$$

$$\bar{t}_j(x) = -t_j(x) \tag{2.8.3b}$$

For a stiff insert in which the elastic modulus is much greater than the modulus of the composite matrix, the Poisson ratio of the insert can be assumed equal to that of the matrix with little error. Therefore, upon consideration of the surface normals at the interface and examination of the F_{ij} kernels, we can write the following relation for the m^{th} insert

$$F_{ij}^{m}(x,\xi) = -F_{ij}(x,\xi) \tag{2.8.3c}$$

Substitution of equations (2.8.3) into equation (2.8.2) yields the following modified boundary integral equation for insert m.

$$C_{ij}^{m}(\xi)u_i(\xi) = \int_{S^m}[-G_{ij}^{m}(x,\xi)t_i(\xi) + F_{ij}(x,\xi)u_i(x)]dS^m(x) \tag{2.8.4}$$

Finally adding N insert equation (2.8.4) to equation (2.8.1) and cancelling terms, yields the modified boundary integral equation for the composite matrix

$$\begin{aligned}\bar{C}_{ij}(\xi)u_i(\xi) = &\int_S [G_{ij}(x,\xi)t_i(x) - F_{ij}(x,\xi)u_i(x)]dS(x)\\ &+\sum_{n=1}^{N}\int_{S^n}[\bar{G}_{ij}(x,\xi)t_i(x)dS^n(x)\end{aligned} \tag{2.8.5}$$

where

$\bar{G}_{ij}(x,\xi) = G_{ij}(x,\xi) - G_{ij}^{m}(x,\xi)$

$\bar{C}_{ij}$ constants dependent on the geometry at ξ

Next,, the two-dimensional kernel integral over the surface of the hole and insert is converted into a line integral. By performing on analytical integration in the circumferential direction on the surface of the hole (or insert) a considerable amount of computational time can be saved in the numerical integration. In this process the holes and inserts are assumed to be circular and a circumferential variation of $a_o + a_1 cos\theta + a_2 sin\theta$ is assumed in the displacements and tractions on the surface of the hole and inserts. Furthermore, tensor transformations on the kernels are necessary for inserts oriented a oblique angles with respect to the global axes. The resulting kernels, contain a large family of elliptical integrals

Once again the analytical integration is complete, the inserts are discretized, assembled and solved as described in Section 3.3.

3.0 NUMERICAL IMPLEMENTATION

The integral equation formulations discussed in the preceding section becomes of practical interest only when numerical techniques are employed for their solution, since analytical solutions exist only for the simplest geometries and loadings. In basic terms, the outline of the numerical solution process is:

1 - Approximate the part geometry and the variation of field quantities (e.g., displacement, traction, temperature) using a suitable set of computationally convenient simple functions.

2 - Evaluate the surface and volume integrals numerically.

3 - Assemble a set of algebraic equations by utilizing the boundary conditions specified for the problem.

4 - Solve the equation system.

5 - Solve for interior results if desired.

In the case of a transient problem (or a quasi-static analysis with multiple load cases) steps 3 and 4 are repeated. In a nonlinear analysis, iteration at each load or step will normally be required in order to satisfy the constitutive equation. The basic steps in the numerical solution process are very briefly discussed in the sections which follow. Full details are available in the sources previously cited [2-13].

3.1 REPRESENTATION OF GEOMETRY AND FUNCTIONS

The method employed in **BEST3D** for the approximation of both part geometry and field quantities is based on the use of the isoparametric shape functions originally developed for use in finite element analysis. The entire boundary of the part is modelled as the union of isoparametric patches. Over each patch the variation of each of the cartesian coordinates is approximated as

$$x_i = N^k(\eta) x_i^k \tag{3.1.1}$$

where the x_i^k are the nodal values of the coordinate and N^k are interpolation functions which take the value 1.0 at a single node and vanish at all others. Continuity along element boundaries is ensured.

In **BEST3D** the variation of (known and unknown) field variables is carried out over each surface patch using the isoparametric shape functions. It is frequently the case that full quadratic variation of the field quantities is not required, so the option of using linear, quadratic or a mixture of linear and quadratic interpolation functions is provided. Geometry is always modelled using the quadratic interpolation functions. Four special modelling capabilities should be noted:

1 - Substructuring capability is provided. This allows a part to be modelled as an assembly of several generic modelling regions (GMR). The GMRs, each of which must be a complete structure, are joined by enforcing appropriate compatibility conditions across common surface patches.

2 - Linear and quadratic variation of field quantities may be mixed within a single GMR.

3 - Infinite elements are provided for use in problems in which boundaries extend to infinity. A decay function is introduced to ensure appropriate behavior of the solution at infinity.

4 - The surface patches can be specialized for fracture mechanics analysis by an appropriate relocation of certain midside geometry nodes.

Volume modelling in **BEST3D** is also based on the use of the isoparametric shape functions. Both volume cell geometry and the variation of field quantities within the cell are mapped using the quadratic isoparametric shape functions. Volume modelling is required in the boundary element method only when thermal loads and/or nonlinear response are involved, and only the portion of the part in which these effects occur must be modelled. Nodes of the volume cells lying on the surface need not match nodes of the surface mesh in either location or node numbering.

An additional volume modelling capability using rectilinear cells in three dimensions is also provided in **BEST3D** . This newly developed capability is presently used for the modelling of embedded holes and for the definitions of hot spots within a structure.

An alternative analysis using the method of particular integrals, with global shape functions, has been developed also for inelastic analysis. This newly developed capability allows the user complete freedom from volume modelling. One merely needs to add some representative interior points where plasticity is likely to occur and isolate that region into a complete GMR. Investigation of its use in nonlinear analysis is continuing.

3.2 SURFACE AND VOLUME INTEGRATION

With the exceptions of volume integrals over rectilinear cells and of jump terms in principal value integrals, all surface and volume integrations in **BEST3D** are done numerically, using product Gaussian quadrature rules. All of the integrals involve the integration, of products of the patch interpolation functions and the kernel functions, over a particular surface patch (or volume cell). In the process of creating the equation system the source point is successively located at each node of the body at which unknowns must be determined. During the calculation of any given integral, the source point may be part of the surface patch (or cell) being integrated (singular case) or not (nonsingular case). Error estimates have been developed to allow determination of appropriate integration orders and element subdivisions for both the singular and nonsingular cases [2,25,30-34]. Similar discussions also apply to the volume integration.

The surface and volume integrations in **BEST3D** require almost no user intervention. Acceptable default values for the few required tolerances have been set internally, based on experience with a variety of linear and nonlinear problems. It should be noted that surface integration in the time embedded dynamic formulation has somewhat different requirements than in the quasi- static case. Reduced integration tolerances may be required for dynamic problems with high forcing frequencies and for transient problems. This essentially allows the program to subdivide the element to appropriately account for oscillatory behavior of the integrand.

3.3 SYSTEM EQUATIONS

The intermediate result of the surface and volume integrations is a set of coefficients which function as multipliers of field quantities (e.g., displacements, tractions, strains, stresses, temperatures, pressures). Some of the field quantities are known from the boundary conditions. During the assembly of the equation system the known and unknown variables are separated and expressed in appropriate local coordinate systems. The coefficients multiplying each set of variables are collected in matrix form for later use. Boundary conditions are imposed, including any required modifications to the coefficient matrices. As a result of these operations, the equations for the entire system, can be written in matrix form as

$$Ax = By + R + C\sigma^{\circ} \tag{3.3.1}$$

and

$$\sigma = A^{\sigma}x + B^{\sigma}y + R^{\sigma} + C^{\sigma}\sigma^{\circ} \tag{3.3.2}$$

where:

x	= vector of unknown displacements and tractions
y	= vector of known displacements and tractions
σ°	= vector of (unknown) initial stress, where applicable
R, R^{σ}	= vectors of body force or source contribution.

Note that equation (3.3.2) is necessary for problems in which interior stresses are required.

In linear periodic analysis R and R^{σ} are assumed to be zero and in transient time-domain analysis these represent the past dynamic history of boundary excitation.

In any substructured (multi-GMR) problems, the matrix A in (3.3.1) contains large blocks of zeros, since separate GMRs communicate only through common surface elements. In order to save both storage space and computer time the matrix is stored in a block basis, with the zero blocks being ignored. The matrices in (3.3.2) are block diagonal, since interior results in any GMR involve only surface and volume integrations relative to that GMR.

In an inelastic analysis the constitutive equations of the form

$$\dot{\sigma}_{ij}(\xi) = D^{ep}_{ijkl}\dot{\epsilon}_{kl} \tag{3.3.3a}$$

with

$$\dot{\sigma}^{\circ}_{ij}(\xi) = (D^{e}_{ijkl} - D^{ep}_{ijkl})\dot{\epsilon}_{kl}$$

defining the initial stress rates for an elastoplastic material and the form

$$\dot{\sigma}^{\circ}_{ij}(\xi) = D^{vp}_{ijkl}\sigma_{kl} \tag{3.3.3b}$$

defining the initial stress rate for a viscoplastic material are necessary to establish the magnitude of the vector σ°.

Two inelastic solution algorithms are contained in **BEST3D** for the solution of vector x (and vector σ^o). These algorithms, which are both incremental procedures, are described below.

Iterative procedure [25] In this procedure equations (3.3.1) and (3.3.2) are used together with equation (33) to solve the system in an incremental fashion. The σ^o vector is initially assumed zero and then is determined in an iterative process during each load increment. The implementation includes a time saving feature that reduces the number of iterations needed for convergence, by utilizing the past history of initial stresses to estimate the value of initial stress rates for the following load increment. The iterative procedure has been implemented for the Von Mises model (with strain hardening), the Two-surface cyclic plasticity model and the thermally sensitive viscoplastic model.

Variable stiffness (direct) procedure [37] This procedure exploits the plastic flow rule to express the unknown initial stress rate tensor as a non-negative scalar quantity $\dot{\lambda}$.

$$\dot{\sigma}^o_{ij} = K_{ij}\dot{\lambda} \tag{3.3.4}$$

and from the other constitutive relations, the current stress rate is related to $\dot{\lambda}$ by

$$\dot{\lambda} = L_{ij}\dot{\sigma}_{ij} \tag{3.3.5}$$

where K_{ij} and L_{ij} are tensor quantities dependent on the current state of the stress.

Equation (3.3.4) is used on a nodal basis to eliminate the σ^o vector from the equations (3.3.1) and (3.3.2). Employing equation (3.3.5) (on a nodal basis), equation (3.3.2) can be back subsituted into equation (3.3.1) yielding the final system equation for a given increment

$$A^*x = B^*y + R^*$$

the x vector is the only unknown and standard numerical techniques are employed for its solution.

This method, in the present version of **BEST3D** , is restricted to the Von Mises model (with strain hardening) under monotonic loading.

3.3.1 ASSEMBLY OF EQUATION FOR COMPOSITE INSERTS

Here, efficiency is of utmost importance. The approach to writing an efficient algorithm is to keep the number of system equations to a minimum by eliminating all unnecessary unknowns from the system. The strategy used is to retain in the system only traction variables on the matrix/ insert interface. This is in contrast to a general multiregion problem where both displacement and tractions are retained on an interface. The elimination of the displacements on the interface is achieved through a backsubstitution of the insert equations into the system equations which are made up exclusively from equations written for the composite matrix (on the outer surface and on the surface of the holes). The procedure is described below.

The boundary integral equation for the composite matrix derived in Section 2.11 is used to generate a system of equations for nodes on the outer surface of the matrix and for nodes on the surface of the holes containing the inserts. Written in matrix notation, we have

On Matrix Surface:

$$\mathbf{Gt} - \mathbf{Fu} + \bar{\mathbf{G}}\mathbf{t}^{\mathbf{H}} = 0 \tag{3.3.6a}$$

On Hole for Insert:

$$\mathbf{Gt} - \mathbf{Fu} + \bar{\mathbf{G}}\mathbf{t}^{\mathbf{H}} = \mathbf{Iu}^{\mathbf{H}} \tag{3.3.6b}$$

where

$\mathbf{t}$ and $\mathbf{u}$ are traction and displacement vectors on the outer surface of the components

$\mathbf{t}^H$ and $\mathbf{u}^H$ are traction and displacement vectors on the hole

$\mathbf{I}$ is the identity matrix

Our goal is to eliminate $\mathbf{u}^H$ from the system. To this end, the integral equation for the inserts derived in Section 2.11 is written for each node on an insert, collocating slightly outside the insert [where $C_{ij}^m = 0$], we obtain

$$\mathbf{F'u^H} = \mathbf{G't^H} \tag{3.7}$$

Pre-multiplying equation (3.3.6b) by the $\mathbf{F'}$ matrix in equation (3.7) yields

$$\mathbf{F'Gt} - \mathbf{F'Fu} + \mathbf{F'}\bar{\mathbf{G}}\mathbf{t^H} = \mathbf{F'u^h} \tag{3.8}$$

Equation (3.7) can now be set equal to equation (3.8) and the final form of the system is derived.

On Matrix:

$$\mathbf{G}\mathbf{t} - \mathbf{F}\mathbf{u} + \bar{\mathbf{G}}\mathbf{t}^{\mathbf{H}} = 0$$

On Hole:

$$\mathbf{F}'\mathbf{G}\mathbf{t} - \mathbf{F}'\mathbf{F}\mathbf{u} + (\mathbf{F}'\bar{\mathbf{G}} - \mathbf{G}')\mathbf{t}^{\mathbf{H}} = 0 \tag{3.9}$$

At every point on the outer surface, either the traction or the displacement is specified and on the surface of the hole only the tractions are retained. Therefore, the number of equations in the system are equal to the final number of unknowns, and hence, the system may be solved. Thereafter, equation (3.36) is used to determine the displacement on the matrix/insert interface.

Since the displacement about a particular hole is present only in the insert equation corresponding to that hole, backsubstitution can be performed one insert at a time in a more efficient manner than backsubstitution of all inserts at once. Further note that nowhere in the assembly process is a matrix inversion necessary. This efficient assembly process was made possible due to the unique formulation of the modified boundary integral equations developed earlier in this section.

When the composite matrix is divided into a multi-region model, the above insert assembly is performed for each region independently. Thereafter, equilibrium and compatibility conditions are invoked at common interfaces of the substructured matrix composite. After collecting together the known and unknown boundary quantities, the final system can be expressed as

$$\mathbf{A}^{\mathbf{b}}\mathbf{x} = \mathbf{B}^{\mathbf{b}}\mathbf{y} \tag{3.10}$$

where

$\mathbf{x}$ is the vector of unknown variables at boundary and interface nodes,

$\mathbf{y}$ is the vector of known variables, and

$\mathbf{A}^{\mathbf{b}}, \mathbf{B}^{\mathbf{b}}$ are the coefficient matrices

Standard numerical procedures can be used to solve the unknowns in equation (3.10). Details are described in the section.

Once all the displacements and tractions are known on the matrix outersurface and on the matrix/insert interface, interior quantities of displacement, stress and strain can be determined at any point in the composite matrix or in the insert. For displacement either the conventional boundary displacement integral equation can be employed or alternatively the modified equations can be used.

Equations for strains are derived from the displacement equations and the strain-displacement relations. Thereafter, equations for stress are obtained by substituting the resulting strain equations into Hooke's law. For points on the boundary, the stress/strain boundary calculation normally employed in boundary elements is used.

3.4 EQUATION SOLUTION

The first step in the process is the decomposition of the system matrix A. In order to minimize the time requirements this is done using the block form of the matrix. The decomposed form is stored for later use. At each load increment or time step all of the known field variables are calculated and multiplied by appropriate coefficient matrices to form the load vector. The decomposed form of the system matrix is then used to calculate the unknown field variables. In a homogeneous, quasistatic linear analysis, this completes the solution, and the process is repeated for the next time step.

The solution process for the time embedded transient algorithm is very similar to the above case, except that the evaluation of the load vector requires a new surface integration at each time step. However, the system matrix remains unchanged from one time step to the next.

In an inelastic analysis two solution algorithms have been implemented: An iterative algorithm [25]; and a (direct) variable [37] stiffness type approach. In the first procedure an iterative process is required in order to satisfy the constitutive relations. This iteration involves use of both the interior stress identity and the constitutive equations. In order to improve convergence of this process, initial stresses for each load increment are extrapolated from past history except at points of load reversal. This reduces the computational costs substantially for most elastoplastic and viscoplastic problems based on the iterative method.

The second approach, known as the variable stiffness method, exploits certain features of the constitutive relationship to express the unknown nonlinear initial stress tensor as a scalar quantity which can be eliminated from the boundary equation system through a back subsituition of the (modified) stress rate equation. Consequently, a new system matrix is generated and must be solved at each load increment.

A new solver was written for **BEST3D** . It operates at the submatrix level, using software from the LINPACK package to carry out all operations on submatrices. The system matrix is stored, by submatrices, on a direct access file. The decomposition process is a Gaussian reduction to upper triangular (submatrix) form. The row operations required during the decomposition are stored in the space originally occupied by the lower triangle of the system matrix. Pivoting of rows within diagonal submatrices is permitted.

The calculation of the solution vector is carried out by a separate subroutine, using the decomposed form of the system matrix from the direct access file. The process of repeated solution, required for problems with time dependent and/or nonlinear behavior, is highly efficient.

3.5 INTERIOR RESULTS

Once all the displacements and tractions are known on the boundary interior quantities of displacement, stress and strain can be determined at any point in or on the body. For displacement the conventional boundary displacement integral equation is employed for point in the interior of the body. The displacements for points on the surface of the body can be calculated by simply using the appropriate shape function for the element on which the point is contained.

Equations for strains are derived from the displacement integral equations and the strain-displacement relations. Thereafter, equations for stress are obtained by substituting the resulting strain equations into the appropriate constitutive equation.

The resulting equations, however, are not only invalid on the surface, but also difficult to evaluate numerically at points close to it. For points on the surface, the stresses can be calculated by constructing a local Cartesian coordinate system with the axes 1 and 2 directed along the tangential directions and the axis 3 in the direction of the outward normal. The stresses $\bar{\sigma}_{ij}$ referred to these local axes (indicated by overbars) are then given by:

$$
\begin{aligned}
\bar{\sigma}_{11} &= \frac{\nu}{1-\nu}\bar{t}_3 + \frac{E\nu}{1-\nu^2}(\bar{\varepsilon}_{11}+\bar{\varepsilon}_{22}) + \frac{E}{1+\nu}\bar{\varepsilon}_{11} \\
\bar{\sigma}_{12} &= \bar{\sigma}_{21} = \frac{E}{2(1+\nu)}\bar{\varepsilon}_{12} \\
\bar{\sigma}_{22} &= \frac{\nu}{1-\nu}\bar{t}_3 + \frac{E\nu}{1-\nu^2}(\bar{\varepsilon}_{11}+\bar{\varepsilon}_{22}) + \frac{E}{1+\nu}\bar{\varepsilon}_{22} \\
\bar{\sigma}_{33} &= \bar{t}_3 \\
\bar{\sigma}_{32} &= \bar{\sigma}_{23} = \bar{t}_2 \\
\bar{\sigma}_{31} &= \bar{\sigma}_{13} = \bar{t}_1
\end{aligned}
\tag{3.13}
$$

where E is the Young's modulus, $\bar{\sigma}_{ij}$ defines the components of the elastic strains in the local axes obtained from the derivative of the shape functions and nodal displacements (minus nonlinear strains) and $\bar{t}_i$ are the traction on the boundary.

3.6 ACCURACY AND EFFICIENCY

Boundary integral formulations for any problem represent the exact statement of the problem posed. In order to maximize the benefits it is essential not to introduce any unnecessary approximations purely for the ease of programming. In this regard, the use of nonconforming elements should be avoided. Although the use of such elements reduces skilled programming effort by nearly an order of magnitude, it becomes virtually impossible to analyze any realistic three-dimensional problem by such a system. Indeed, a pilot study to compare the relative efficiency and accuracy of conforming and nonconforming elements shows that conforming elements are more accurate for the same number of quadratic elements. Additionally, computing costs are much higher for nonconforming elements [30]. For example, with 6, 10, 14 and 18 elements the nonconforming element consumed respectively 4, 5, 7 and 10 times more CPU time than conforming elements. It should be noted that the early work of Lachat and Watson [31,32] on conforming elements provided the inspiration for the present implementation.

4.0 GUIDE TO USING BEST3D

Since **BEST3D** has been designed to cover a very wide range of problems in engineering it may appear to be a little difficult for a beginner to get started. This section is therefore written to provide some guidance to such an user.

4.1 GETTING STARTED

Usually the user is motivated by a specific problem in a given technical area (heat transfer or elasticity for example). It is suggested that he should first read the analysis section of the manual to get some flavor of the BEM in that area. He should then briefly examine the structure and organization of the input data in Section 5 in conjection with a sample problem data set given in Section 6. Section 7 may contain a brief description of a specific engineering example in the technical area of interest to the user.

It may also be helpful to use a specific test data given in Section 6 and modify it to create a new test problem. In order to do this the user must of course study the relevant parts of Section 5.

4.2 AVAILABILTY OF BEST3D

BEST3D is written in FORTRAN 77 and is therefore adaptable to any computer which uses such a compiler. Since most of the development work has been carried out on IBM and CRAY mainframes as well as HP-9000 and SUN-4 desktop workstations, it is most easily ported to these systems.

4.3 FILESYSTEM IN BEST3D

BEST3D makes use of unit 5 as its input data file and unit 6 as its output file. In addition to these an extensive set of disk files are used during the execution of the code. For the complete range of analysis used in **BEST3D** it is necessary to have the 60 simultaneous open files in the system. Not all of these files are necessary for the simpler analyses such as quasistatic elastic or steady state heat transfer analyses, where usually only 1/3 of the total are used. For more complex time dependant analyses extensive use of disc files is made. The files are either of sequential or direct access type and are defined as FT** based on IBM terminology.

In the current version of **BEST3D** the volume of data stored on disc during the execution of the code is high. This fact is rather critical to a workstation based user who must have atleast 300MB user area to run any reasonable practical problem. It is proposed to develop a more efficient disc usage strategy in the next release of **BEST3D** .

4.4 SPECIAL FEATURES OF BEST3D

4.4.1 DEFINITIONS

The following definitions are used throughout the manual.

Points, Nodes or Nodal Points - are generic names for all points in a data set for which coordinates are defined. These points may be source points and/or geometric points which are used in the boundary, volume, hole and insert discretizations, or they may be used to define a point of a thermal hot spot or a sampling point. Points are usually user defined, however, **BEST3D** may create additional points through automated surface generation (see the argument REFNAME on the SURF card in **GMR input in chapter 5). Additional Nodes may also be generated by **BEST3D** for hole and/or inserts. All points defined in a data set by the user should have unique node numbering.

Geometric Points - are points used in the geometrical definition of the body of interest. Specifically, geometric points are used in the description of the geometry of a boundary element, hole element, insert element, or volume cell. Geometric points also may or may not be source points.

Source Points - refers to boundary source points, or boundary and volume source points in an analysis. Source points are used in the functional representation of variables across a boundary element, across a volume cell or in a global shape function representation. In a system equation, unknowns are retained at source points.

Functional Nodes - same as source points

Boundary Source Points - are points in a discretization of the boundary surface (or interface) which are used in the functional representation of the field variables across the boundary elements. At every boundary source point (and only at boundary source points) unknowns in the boundary system equation are retained corresponding to the unknown boundary conditions at these points. Likewise, known boundary conditions (implicitly or explicitly defined) are required at these points. Boundary conditions specifications for points other than boundary source points will result in a fatal error. Boundary source

points are selected by **BEST3D** based on the type of functional variation of the primary variables across the boundary element which is defined in the data set by the user. (see SURF and TYPE cards under **GMR input in Chapter 5)

Volume Source Points - are points in a volume discretization which are used to represent the functional variation of certain variables through the volume of the body via volume cells or a global shape function variation. These are required only in nonlinear analysis or when the body is subjected to certain types of body forces. In the case of nonlinear analysis, unknowns are retained at volume source points which have to be solved for, along with the unknowns at the boundary source points. This entails writing additional equations at each volume source point. In the case of body forces, the variables are known quantities and additional equations are therefore unnecessary. Volume source points are selected by **BEST3D** based on the type of functional variation selected by the user. For the volume cell approach see VOLU and TYPE cards under **GMR input in Chapter 5. For the global shape function approach, volume source points are taken at every node defined in the global shape function (see GLOB and NODE cards under **GMR input in Chapter 5.)

Sampling Points - are (user defined) points in the interior of the body or on the surface of the body for which results are requested. Results at sampling point are calculated after the system equation is solved. Sampling points are input on a separate list (see SAMP card in **GMR input in Chapter 5) and are totally independent of the point list used for boundary and volume discretization and independent of the point list for holes and inserts. A sampling point may coincide with a boundary and volume discreizations point (but not with a hole or insert). Sampling points should use unique node numbering.

Hole Elements - are curvilinear line elements in three dimensional space that are used to represent a surface of a tubular shaped hole in a 3-D body. The line element represents the centerline of the hole and the surface of the hole is at a prescribed radius from the centerline. Basically, the hole element is a computationally efficient and convienient way to model a tubular hole in a 3-D body.

Insert Elements - are used to model a circular composite fiber in a 3-D body where the elastic modulus of the fiber is different from that of the matrix. The insert element is modelled similar to a hole element except, instead of a void, the material of a prescribed modulus is assumed to fill the void.

Volume Cells - Certain analyses require an integration of some variable over all or part of the volume of the body. In this cases the volume is divided into smaller parts called volume cells, where interpolation functions (of some order) are used to represent the variation of the variable to be integrated across the volume cell. In some analyses a global shape function may be used as an alternative to volume cells.

Global Shape Functions - in certain analyses a volume integral is present in the governing boundary integral equation. Instead of using an expensive volume integration across the body (via volume cell discretization) the problem can be convienently recast using the particular integral approach to eliminate the presence of the volume integral. If this approach is used and if the variable involved in the particular integral is allowed to assume an arbitrary variation across the body (or subregion) then a global shape function is used to model this variation through the body. The global shape function is a general interpolation function with an arbitrary number of volume source points located at any (reasonable) random position. (see GLOB under **GMR input in chapter 5)

Generic Modeling Region (GMR) - in a boundary element analysis the body under investigation may be fictitously divided in a number of smaller parts for convienence in mesh modelling and efficiency in computation. Each part is called a generic modelling region and is modelled as an individual boundary element model. The nodes and elements of each region must match up at common interfaces and are connected by relations defined by the user. The term generic refers to the fact that only geometry is defined in this section and the part defined can then be used for any type of analysis (e.g. elasticity, heat transfer etc.)

4.4.2 MESH SIZE

Most of the currently available experience of developing mesh for a given problem is based on nearly two decades of the finite element or finite difference analyses. It is possible to take only the boundary part of a given finite element mesh system to generate the boundary element mesh system. Unfortunately this often leads to an inefficient BEM analysis because of use of too many unnecessary elements. For three-dimensional problems it is often necessary to reduce this residual surface elements so generated by almost a factor of two to four. If the geometry of the surface is too complex so that this reduction could not be achieved without sacrificing geometrical details that are important in the analysis then one must use the mesh without modification.

4.4.3 SPECIFICATION OF BOUNDARY RESTRAINTS

BEM analyses require the same type of restraint on the boundaries so that unwanted movements of the solution region is prevented. It must be noted however, restraints often introduces local singularities, particularly at the edges of the restraints. Since finite element method does not capture any of these local singularities, engineers do not have to pay any special attention to them.

Since the specification of the boundary element condition needs to be very precise in a boundary based solution scheme it is often possible to capture high stress and strain gradients purely by specifying unnecessary restraints. For example a cantilever completely fixed at the supported end by specifying both displacement components zero will not provide the correct distribution of bending and shear stresses of a beam solution at the supported end. A suitable roller support, on the other hand, will provide the beam bending solution.

It is therefore strongly recommended that a user specifies roller support to simulate the necessary restraints.

4.4.4 PROBLEMS OF EDGES AND CORNERS

In the case of a potential flow problem the potential is uniquely defined but its normal derivatives are multivalued at a corner node. Similarly, for an elasticity problem the displacements are uniquely defined but the surface tractions are multivalued at a corner node. Thus if we wish to write the equation (for an elasticity problem)

$$\beta \mathbf{u}^{\mathrm{P}} = \sum_{\mathrm{q}=1}^{\mathrm{N}} \left[\left(\int_{\Delta \mathrm{S}} \mathrm{G}^{\mathrm{Pq}} \mathrm{N}^{\mathrm{q}} \mathrm{ds} \right) \mathbf{t}_{\mathbf{n}} - \left(\int_{\Delta \mathrm{S}} \mathrm{F}^{\mathrm{Pq}} \mathrm{N}^{\mathrm{q}} \mathrm{ds} \right) \mathbf{u}_{\mathbf{n}} \right] \qquad 4.1$$

for m boundary nodes including one true corner node, the resulting final system of equations will be

$$\mathbf{Gt} - \mathbf{Fu} = 0 \qquad 4.2$$

where
$\mathbf{F} =$ $3m \times 3m$ matrix for three-dimensional problems

and
$\mathbf{G} =$ $3m \times (3m+4)$ matrix for three-dimensional problems

The additional columns in $\mathbf{G}$ arise from the multivalued tractions defined at the corner node. If these tractions are all specifically prescribed then the solution of Eq. (4.2) presents no difficulty. A suitable mixture of tractions and displacement boundary conditions (Fig. 4.1a-b) at the corner also presents no difficulty if the final system matrix involving all the unknowns is square and of order $3m \times 3m$ for three-dimensional problems. If the displacements alone are specified at the corner (Fig. 4.1c) it is not possible to solve Eq. (4.2) and an alternative approach must be found.

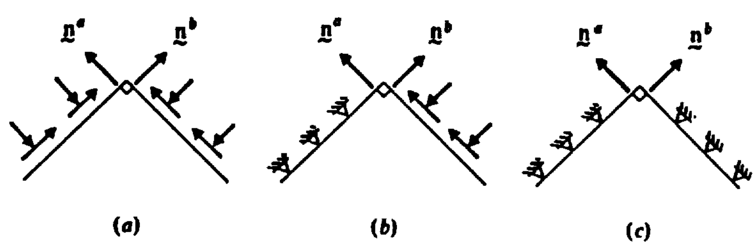


Fig. 4.1 The problems at a corner of the boundary

It should be noted that the preceding remarks only apply to a sharp corner that actually exists in a specific problem. The discretization of a smooth surface using flat boundary elements also results in boundary discontinuities, but these must be treated as if the boundary were continous if correct results are to be obtained.

5.0 BEST3D INPUT

The basic input required by **BEST3D** is the definition of **Geometry, Material Properties and Boundary conditions.** While this is the same definition required by a finite element structural analysis program, a somewhat different set of information is required to accomplish the definition for a boundary element program.

The input to **BEST3D** is intended to be as simple as possible, consistent with the demands of a general purpose analysis program. Meaningful keywords are used for the identification of data types. Free field input of both keywords and numerical data is permitted, however there are a number of general rules that must be followed.

General Rules for Input Data

1. Upper Case

All alphanumeric input must be provided in upper case.

Proper Usage:

```
**CASE
  TITLE  TRANSIENT ELASTODYNAMICS - TEST CASE
  TRANSIENT 8 0.5
  SYMMETRY QUARTER
```

Improper Usage:

```
  transient 8 0.5
  SYMMETRY quarter
```

2. Parameter Positioning

Parameters may appear anywhere on an input line, as long as they appear in the proper order and are separated by at least one blank space.

Proper Usage:

```
CTHERMAL PLASTICITY STEADY 5 1.0
ELEMENT 1 6 8
```

Improper Usage:

```
CTHERMALPLASTICITY STEADY 5 1.0
PLASTICITY UTHERMAL 5 STEADY 1.0
```

General Rules for Input Data

3. Keyword Truncation

Any keywords that are longer than four letters may be truncated to the first four letters.

Proper Usage:

```
SYMMETRY QUAR
SYMM QUAR
ELEM 1 6 8
```

Improper Usage:

```
SYMMETRY QUA
```

4. Floating Point Numbers

Any real parameters may be input in either FORTRAN E or F format. In the latter case, an Fn.0 is assumed with $n \leq 8$.

Proper Usage:

```
EMOD 30.E+6
ALPHA 1.E-06
POINTS
  0.004 1.110 0.0
```

Improper Usage:

```
EMOD 30000000.0
ALPHA 1.-6
POINTS
  4.0-3 1.110 0
```

General Rules for Input Data

5. Comments

Comments can be inserted in the data file by placing a dollar sign ($) anywhere on an input line. The remainder of that input line is then ignored by the **BEST3D** input processor.

Proper Usage:

```
ELEMENT 1 6 8 $ ELEMENTS ON THE OUTER RIM
$
$ MODIFIED 03/08/88 GFD
  POINTS 25 26 27
```

6. Blank Lines

Blank Lines can be inserted anywhere in the data file and are useful for aesthetic purposes.

7. Units

A consistent system of units must be used for input of all types (material properties, geometry, boundary conditions, time steps). Output will be in the same consistent system of units. The selection of appropriate units is the user's responsibility.

8. ** Keywords

Certain keywords are prefixed by the symbol **. These identify the beginning of a block of data of a particular type, and serve to direct the program to the appropriate data processing routine. There should be no blank spaces between the ** symbol and the pertinent keyword. Additionally, the ** data blocks must appear in the following specific order:

**CASE
**MATERIAL
**GMR
**INTERFACE
**BCSET
**BODY
**BCCHANGE

There may be multiple data blocks of each type, except for the **CASE block.

General Recommendations for Input Data

1. Ordering of Input Items

While there is some flexibility in the ordering of lines within **BEST3D** data set, it is strongly recommended that the user follow the order provided in the manual. Examples of proper ordering are provided throughout this chapter.

2. Documenting Data Sets

The **$** keyword is provided to permit comments anywhere in the input data set. This should be used generously to fully document the analysis. Blank spaces can also be used to improve readability. The format, displayed in the examples of this chapter, is recommended.

General Limits of BEST3D

It should be noted that there are certain limits which must be observed in the preparation of input for **BEST3D** . These limits are of two main types:

1 - Limits on the maximum number of entities of various types within a single analysis.

2 - Limits on the user specified numbering of certain entities.

The present limits are summarized below. It is anticipated that certain of these limits may be relaxed in future versions of **BEST3D.**

ENTITY*	LIMIT
total source points	1200
total elements (including holes)	600
total volume source points	600
generic modelling regions (GMRs)	15
surfaces per GMR	15
source points per GMR	600
elements (including holes) per GMR	300
holes per GMR	100
infinite elements per GMR	11
interface and cyclic symmetry boundary condition sets	20
element pairs (interfaces and cyclic)	99
node pairs (interfaces and cyclic)	350
spring boundary condition sets	60
time points in boundary condition sets	20
user specified element number	9999
user specified node number	9999

* Definition of the terminology used in this table can be found in Section 4.4.

Individual Data Items

The remainder of this chapter provides detailed information on each of the data items available within **BEST3D**. The individual items are grouped in sections, under the associated ** keyword, as follows:

5.1 CASE CONTROL INFORMATION (**CASE)
5.2 MATERIAL PROPERTY DEFINITION (**MATE)
5.3 GEOMETRY DEFINITION (**GMR)
5.4 INTERFACE DEFINITION BETWEEN SUBREGIONS (**INTE)
5.5 BOUNDARY CONDITION DEFINITION (**BCSE)
5.6 BODY FORCE DEFINITION (**BODY)
5.7 BOUNDARY CONDITION CHANGE DEFINITION (**BCCH)

5.1 CASE CONTROL

This input section provides **BEST3D** with information controlling the overall execution. It provides the title and determines which of the major program branches will be executed. It also defines the times at which solutions of the given problem are to be evaluated. This section must be input exactly once for each analysis and must be input before any other data.

A list of keywords recognized in the case control input are given below, and a detailed description follows.

SECTION	KEYWORD	PURPOSE
5.1.1 Case Control Input Card		
	**CASE	Start of case control input
5.1.2 Title		
	TITL	Title of job
5.1.3 Times for Output		
	TIME	Times of solution output
5.1.4 Type of Analysis		
	ELAS	Elastic analysis
	FORC	Forced vibration analysis
	FREE	Free vibration analysis
	HEAT	Heat Transfer analysis
	PLAS	Plasticity analysis
	TRAN	Transient elastodynamic analysis
5.1.5 Geometric and Loading Symmetry Control		
	SYMM HALF	Symmetry about Y-Z plane
	SYMM QUAR	Symmetry about X-Z and Y-Z planes
	SYMM OCTA	Symmetry about X-Z, Y-Z, and X-Y planes

SECTION	KEYWORD	PURPOSE
5.1.6 Special Loading Control		
	LOAD COMP	Complex-valued boundary conditions in periodic dynamic analysis
5.1.7 Restart Facility		
	REST WRIT	Save integration files for future runs
	REST READ	Use integration files from previous run
5.1.8 Integration Precision and Efficiency Optimization		
	PREC	Numerical integration precision
5.1.9 Output Options		
	ECHO	Produce echo of input data
	PRIN BOUN	Printout displacement and traction results
	PRIN NODA	Print boundary displacement, stress, strain at nodal points
	PRIN LOAD	Print load calculation

5.1.1 CASE CONTROL INPUT CARD

**CASE

Status - REQUIRED

Full Keyword - `**CASE` control

Function - Identifies the beginning of the case control input section.

Input Variables - NONE

Additional Information - NONE

Examples of Use -

1. Request a three-dimensional steady-state heat transfer analysis.

```
**CASE
  TITLE  HEAT CONDUCTION IN A MOLD
  HEAT
```

5.1.2 TITLE

TITL CASETITLE

Status - REQUIRED

Full Keyword - TITLE

Function - Defines title for analysis.

Input Variables -

CASETITLE (Alphanumeric) - REQUIRED - 72 chars. max. length

Additional Information - NONE

Examples of Use -

1. Describe the analysis.

```
**CASE
  TITLE  TURBINE BLADE A7311 - THERMOELASTIC ANALYSIS
  ELASTIC
```

5.1.3 TIMES FOR OUTPUT

`TIME    T1 T2 T3 ...  TN`

Status - OPTIONAL

Full Keyword - `TIMES`

Function - Identifies times at which output is required (only for static analysis).

Input Variables -

T1 (Real) - REQUIRED

T2 ... TN (Real) - OPTIONAL

Additional Information -

This input may be continued on more than one card, if required. Each card must begin with the keyword TIME. A maximum of twenty output times may be selected. A minimum of one output time must be chosen.

This card is only functional for static analysis.

Examples of Use -

1. Conduct an elastic analysis at times 1.0, 2.5 and 6.0 and output the results.

```
**CASE
  TITLE  ROTOR - ELASTIC ANALYSIS
  TIMES 1.0 2.5 6.0
  ELASTIC
```

5.1.4 TYPE OF ANALYSIS

ELAS

Status - OPTIONAL

Full Keyword - ELASTIC

Function - Identifies an elastic analysis.

Input Variables - NONE

Additional Information -

Elastic analysis is the default analysis type. Therefore, if none of the keywords from Section 5.1.5 are present, an elastic analysis will be performed.

Examples of Use -

1. Request an elastic stress analysis of a concrete reactor pressure vessel.

```
**CASE
  TITLE  REACTOR PRESSURE VESSEL - LOAD CASE 2B
  ELASTIC
```

FORC W1 W2 W3 ... WN

Status - OPTIONAL

Full Keyword - FORCED

Function - Identifies a steady-state forced vibration analysis.

Input Variables -

W1 (Real) - REQUIRED

Defines the frequency (in rad/time) of the sinusoidal forcing function at which output is required.

W2 .. WN (Real) - OPTIONAL

Additional Information -

This input can be continued on more than one card, if required. Each card must begin with the keyword FORC. A maximum of twenty forcing frequencies may be selected for analysis. A minimum of one forcing frequency must be input.

Examples of Use -

1. Determine the response of an axle driven by loads at frequencies of 15., 30. and 60.

```
**CASE
  TITLE  AXLE - STEADY-STATE FORCED RESPONSE
  FORCED 15. 30. 60.
```

FREE NMOD CTYPE

Status - OPTIONAL

Full Keyword - FREE-VIBRATION

Function - Identifies a free-vibration analysis.

Input Variables -

NMOD (Integer) - OPTIONAL

Sets the number of eigen frequencies required, starting from the lowest frequency. Default is 5.

CTYPE (Alphanumeric) - OPTIONAL

Allowable value is UNCOnstrained. It identifies a problem geometry with unconstrained boundaries.

Additional Information - NONE

Examples of Use -

1. Determine the first four modes of free vibration of a turbine blade.

```
**CASE
  TITLE  TURBINE BLADE A7311 - MODE SHAPES
  FREE 4
```

HEAT ITYPE NSTEP DELTAT

Status - OPTIONAL

Full Keyword - HEAT

Function - Identifies a heat conduction analysis.

Input Variables -

ITYPE Alphanumeric - OPTIONAL

Allowable value is STEA or TRAN. (Default is steady-state analysis)

STEAdy - Identifies a steady-state analysis.

TRANsient - Identifies transient analysis.

NSTEP (I) - REQUIRED for transient problems

Sets the number of time steps for which the transient heat conduction analysis is to be carried out.

DELTAT (FP) - REQUIRED for transient problems.

Defines the time step size.

Additional Information -

In the present version, only a constant time step DELTAT is permitted.

Examples of Use -

1. Find the steady-state temperature distribution in a heat exchanger.

```
**CASE
  TITLE  TUBE-AND-FIN HEX - STEADY-STATE
  HEAT
```

PLAS TYPE DLOAD EPSNORM

Status - OPTIONAL

Full Keyword - PLASTICITY

Function - Identifies an analysis involving nonlinear material response.

Input Variables -

TYPE (Alphanumeric) - OPTIONAL

Allowable values are ITER or DIRE. Default is ITER.

ITER - Iterative plasticity algorithm - initial stress is found through an iterative procedure.

DIRE - Direct plasticity algorithm - (modified) stress equations are back substituted into the boundary system to eliminate the unknown initial stresses (for mono-tonic loading only and homogeneous material only).

DLOAD (Real) - OPTIONAL - (default = .05)

Sets the maximum permitted stress change, as a fraction of the material yield stress, during a single load increment. This variable determines the size of the internal load increment used in **BEST3D**.

EPSNORM (Real) - OPTIONAL - (default = .005)

Tolerance used to determine convergence of the nonlinear algorithm at a particular load step. For convergence, the sum of the changes in the initial stress divided by the yield stress must be less than EPSNORM.

Additional Information -

The variable DLOAD must be input if EPSNORM is input.

If this card is input, an appropriate volume cell discretization or Particular Integral (Nodal) representation must be defined in the geometry input section, described later.

Examples of Use -

1. Perform elastoplastic analysis of a pressure vessel utilizing the direct plasticity algorithm.

```
**CASE
  TITLE  PRESSURE VESSEL - PLASTICITY
  TIMES  1.0  2.0  3.0  4.0
  PLASTICITY DIRECT
```

TRAN ITYPE NSTEP DELTAT

Status - OPTIONAL

Full Keyword - `TRANSIENT`

Function - Identifies a transient elastodynamic analysis.

Input Variables -

ITYPE (Alphanumeric) - OPTIONAL

Allowable value is LAPLace. Default is time domain transient dynamic analysis.

LAPLace - Identifies Laplace domain transient dynamic analysis.

NSTEP (Integer) - REQUIRED

Sets the number of time steps for which the transient analysis is to be carried out.

DELTAT (Real) - REQUIRED

Defines the time step size. Time step size should be DELTAT $\leq \frac{L}{C_p}$, where C_p is the pressure wave velocity and L is the width of the smallest element.

Additional Input -

TIME VARI CTYPE

Status - REQUIRED (if TRAN is input for a time domain analysis)

Full Keyword - `TIME VARIATION`

Function - Identifies type of temporal variation of displacements and tractions.

Input variables -

CTYPE (Alphanumeric) - REQUIRED

CONStant - constant variation of field quantities in time.

LINEar - linear variation of field quantities in time.

Examples of Use -

1. Conduct a transient elastodynamic analysis of a spherical tank using a linear time variation of field variables.

```
**CASE
  TITLE  SPHERICAL TANK - SUDDEN PRESSURIZATION
  TRANSIENT  10  0.01
  TIME VARIATION LINEAR
```

5.1.5 GEOMETRIC AND LOADING SYMMETRY CONTROL

SYMM STYPE

Status - OPTIONAL

Full Keyword - SYMMETRY

Function - Identifies a problem with geometric and loading symmetry.

Input Variables -

STYPE (Alphanumeric) - REQUIRED

Allowable values are HALF, QUAR, and OCTA.

HALF - Half symmetry, about the Y-Z plane.

QUAR - Quarter symmetry, about the X-Z and Y-Z planes.

OCTA - Octal symmetry, about the X-Z, Y-Z, and X-Y planes.

Additional Information -

To model the problem geometry, in all cases, use the part of the geometry which is on the positive side of the axis (axes) of symmetry.

If the SYMM card is used the plane of symmetry does not have to be modelled, and therefore, boundary elements should not appear on the plane of symmetry.

The use of the SYMM card automatically invokes the condition of zero displacement (and zero flux) on and perpendicular to the plane of symmetry. Therefore displacement (and/or flux) in the perpendicular direction does not have to be set to zero at the plane or at any other point for the purpose of preventing (arbitrary) rigid-body motion (in this direction) as is usually required.

Symmetry is not available for problems using Global Shape Function discretization of the volume.

In a problem with anisotropic material, the material must also be symmetric with respect to the symmetry imposed using the SYMM card.

Examples of Use -

1. Perform an elastic analysis on a hollow sphere uitlizing a model of only the first (positve) octant.

```
**CASE
  TITLE  HOLLOW SPHERE WITH INTERNAL PRESSURE
  ELASTIC
  SYMMETRY OCTAL
```

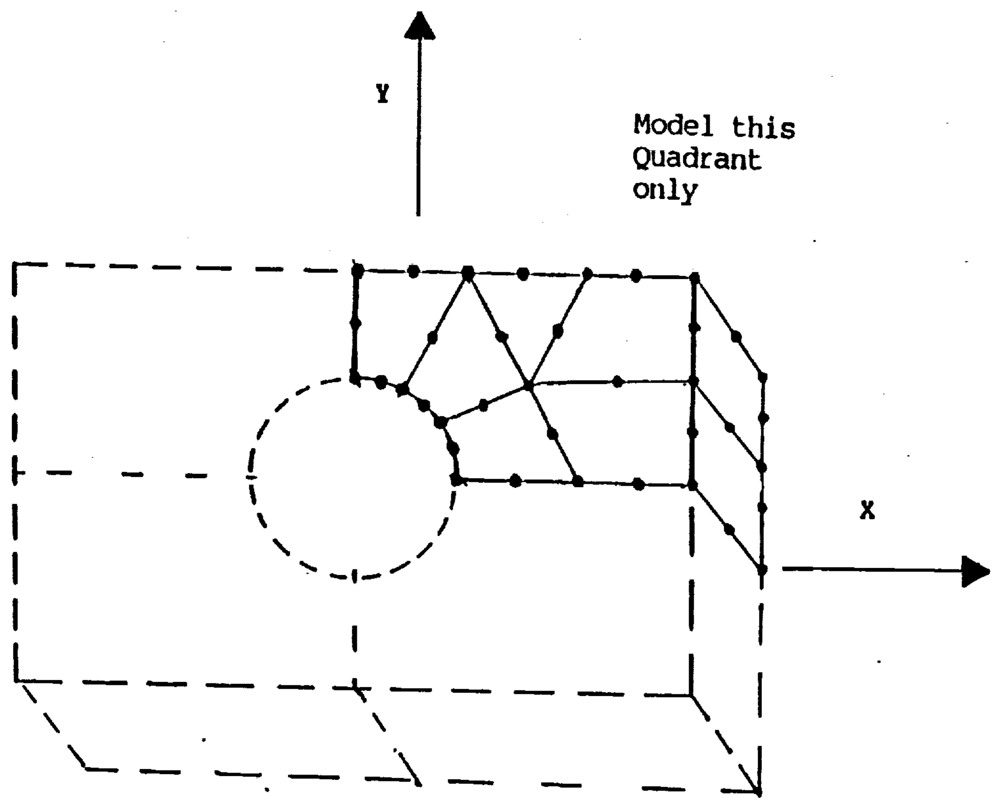


Figure for **CASE: SYMM card
Two-dimensional Quarter symmetry model

5.1.6 SPECIAL LOADING CONTROL

LOAD TYPE

Status - OPTIONAL

Full Keyword - LOADING

Function - Identifies a problem having special loadings (boundary conditions)

Input Variables -

TYPE (Alphanumeric) - REQUIRED

Allowables values are COMP, SYMM

COMP - Complex-valued loadings (boundary condition) for periodic elastodynamic analysis only

Additional Information -

When the control card "LOAD COMP" is used in periodic dynamics, all the boundary conditions have to be input with both real parts and imaginary parts.

Examples of Use -

1. Specify a complex valued loading and displacement boundary condition for periodic elastodynamic (Forced vibration) analysis. The boundary condition set must specify the real and imaginary parts for both loads and displacements.

```
**CASE
  TITLE EMBEDDED STRIP FOOTING
  SYMMETRY HALF
  FORCED .001 0.9 1.5
$ COMPLEX-VALUED LOADING AND DISPLACEMENT B.C
  LOADING COMPLEX
```

5.1.7 RESTART FACILITY

REST RTYPE

Status - OPTIONAL

Full Keyword - `RESTART`

Function - Enables the restart facility for integration.

Input Variables -

RTYPE (Alphanumeric) - REQUIRED

Allowables values are WRIT, READ

WRITe - Saves all of the integration files generated during the current run for later reuse.

READ - Bypasses the integration phase for the current run. Instead, the integration files from a previous run are utilized.

Additional Information -

Integration is generally the most expensive part of any boundary element analysis. Consequently, when the same model is to be run with several sets of boundary conditions, the restart facility should be used.

A complete analysis must first be run with RESTart WRITe specified. The files FT31, FT32, FT33, FT34, FT35, FT36, FT37, FT38 and FT39 are then retained after completion of the run. These files contain all the integration coefficients that were computed. Subsequent runs can then be made, with different sets of boundary conditions, by using RESTart READ. In this case, the integration phase will be skipped. Instead; the integration coefficients will be read from the files FT31, FT32, FT33, FT34, FT35, FT36, FT37, FT38 and FT39.

When employing RESTart READ it is the user's responsibility to ensure that the proper integration files exist.

Geometry and material properties must be the same for both the RESTart WRITe and RESTart READ data sets. However, no checking is done by **BEST3D**. This is the user's responsibility.

The restart facility is not available for transient analyses.

Examples of Use -

1. Save the integration files generated during an elastic analysis of an axle.

```
**CASE
  TITLE  AXLE - LOAD CASE 1A
  TIMES  1.0
  ELASTIC
  RESTART WRITE
```

2. Rerun an elastic analysis of the same axle with a different set of boundary conditions by using existing integration files.

```
**CASE
  TITLE  AXLE - LOAD CASE 1B
  TIMES  1.0
  ELASTIC
  RESTART READ
```

5.1.8 INTEGRATION PRECISION

PREC PTYPE

Status - OPTIONAL

Full Keyword - PRECISION

Function - Provides numerical efficiency through the use of lower order surface integration.

Input Variables -

PTYPE (Alphanumeric) - OPTIONAL

Allowable values are HIGH, MEDIUM, and LOW.

HIGH - Use 4 X 4 integration on each element subsegment.

MEDI - Use 3 X 3 integration on each element subsegment.

LOW - Use 2 X 2 integration on each element subsegment.

Additional Information -

In many cases, MEDIum or LOW order integration produces results that are acceptable for engineering analysis, and significant computational savings can be realized. However, for other problems, particularly those involving thin-walled bodies, PRECision HIGH is recommmended.

High precision (4 X 4 integration) is used if a PREC card is not present.

This card does not affect the integration of hole elements.

Examples of Use -

1. Perform a steady-state heat conduction analysis of a casting mold.

```
**CASE
  TITLE  MOLD COMPONENT 6 - STEADY CONDITIONS
  HEAT
  RESTART WRITE
  PRECISION LOW
```

5.1.9 OUTPUT OPTIONS

ECHO

Status - OPTIONAL

Full Keyword - ECHO

Function - Requests a complete echo print of all card images in the input data set.

Input Variables - NONE

Additional Information - Default is no echo print.

Examples of Use -

1. Request an elastic analysis with an echo of the input data set.

```
**CASE
  TITLE  PRESSURE VESSEL
  ELASTIC
  RESTART WRITE
  ECHO
```

PRIN PTYPE

Status - OPTIONAL

Full Keyword - PRINTOUT-CONTROL

Function - Requests specific printed output.

Input Variables -

PTYPE (Alphanumeric) - REQUIRED

Allowable values are BOUN, NODA, LOAD, ALL, and INTE.

- BOUN - For printing the displacements and tractions, or corresponding quantities such as temperature, pressure, and flux at all boundary source points
- NODA - For printing the displacements, stresses, and strains at all geometry nodes on the boundary. (available only for linear isotropic elasticity)
- LOAD - For printing the resultant load value on each boundary element and the total load equilibrium of each region. (excluding resultant body force)
- ALL - For printing BOUN, NODA, and LOAD information with a single request

Additional Information -

For printing two or more types of output, a separate PRIN request must be included for each type.

If a PRIN, BOUN, NODA or LOAD request does not appear in the case control input then all three types of output (BOUN, NODA, and LOAD) will be printed by default.

Results for sampling points and volume source points are always printed.

Examples of Use -

1. In the elastic analysis of a rotor, print out the resultant boundary element loads.

```
**CASE
  TITLE  ROTOR - ELASTIC ANALYSIS
  TIMES  1.0   2.5   6.0
  ELASTIC
  PRINT LOAD
```

5.2 MATERIAL PROPERTY DEFINITION

This input section defines the linear and, when required, the nonlinear material properties of the various materials used in an analysis. A complete set of material property input must be provided for each material used. At least one set must be input for every analysis. A consistent set of units must be used for all properties.

A list of keywords recognized in the Material input are given below and a detailed description follows.

<u>**SECTION**</u>	<u>**KEYWORD**</u>	<u>**PURPOSE**</u>
5.2.1 Material Property Input Card		
	**MATE	Beginning of a material property input set
5.2.2 Material Identification		
	ID	Identifier of a material type
	LIBR	material library
5.2.3 Mass Parameter		
	DENS	material mass density
5.2.4 Isotropic Elastic Parameters		
	EMOD	Young's modulus
	POIS	Poisson's ratio
5.2.5 Isotropic Thermal Parameters		
	COND	conductivity of material
	SPEC	specific heat
5.2.6 Isotropic Temperature-dependent Thermoelastic Parameters		
	TEMP	temperature values at which elastic material properties will be defined
	EMOD	Young's modulus
	ALPH	co-efficients of thermal expansion

SECTION	KEYWORD	PURPOSE
5.2.7 Anisotropic Elastic Parameters		
	ANIS	identifier for anisotropic material
	STIF	material stiffnesses
	COMP	material compliances
	TECH	Technical constants for an orthotropic material
	ORIE	orientation of the axis of transverse isotropy in 3-D, or, of one material axis with respect to the corresponding geometric axis for anisotropy in 2-D
5.2.8 Isotropic Viscous Parameters		
	DAMP	viscous damping co-efficient
5.2.9 Additional Elastoplastic and Viscoplastic Parameters		
	INEL	signals the beginning of an inelastic material model input
	TIME	time for reversal of loading
5.2.9.1	Von Mises Model	
	VON	Von Mises material model
	YIEL	proportional limits of linear elastic behavior
	CURV	stress-strain curve for Von Mises model with isotropic hardening
5.2.9.2	Two Surface Model	
	TWO	two-surface material model
	YIEL	proportional limits of linear elastic behavior
	HARD	inner and outer proportional limits
5.2.9.3	Walker's Model	
	WALK	Hastaloy-X viscoplastic material model

Note: Refer to the following table for a list of required material properties corresponding to a particular type of analysis.

A list of material properties required for different types of analysis are defined below:

REQUIRED MATERIAL PROPERTIES

TYPE OF ANALYSIS	**MATERIAL PROPERTIES**
1. Isotropic Elastic Stress Analysis	EMOD, POIS (TEMP: optional) (ALPH: if thermal body force is present) (DENS: if centrifugal body force is present) (DENS: if inertial body force is present)
2. Dynamic Analysis:	
2a. Free-Vibration Analysis	EMOD, POIS, DENS (TEMP: optional)
2b. Steady-state or Laplace Domain Transient Analysis	EMOD, POIS, DENS, DAMP (TEMP: optional)
2c. Time Domain Transient Dynamic Analysis	EMOD, POIS, DENS (TEMP: optional)
3. Anisotropic Elastic Stress Analysis	STIF (or COMP or TECH), ORIE (DENS: if centrifugal body force is present) (DENS: if inertial body force is present)
4. Plasticity Analysis:	
4a. Von Mises model	EMOD, POIS, YIEL, CURV (TEMP: optional) (ALPH: if thermal body force is present) (DENS: if centrifugal body force is present) (DENS: if inertial body force is present)

TYPE OF ANALYSIS	MATERIAL PROPERTIES
4b. Two surface model	EMOD, POIS, YIEL, HARD (TEMP: optional) (ALPH: if thermal body force is present) (DENS: if centrifugal body force is present) (DENS: if inertial body force is present)
4c. Viscoplastic (Walker) model	Material Library ID Name
5. Heat Transfer Analysis	
5a. Steady-state (Potential Flow)	COND
5b. Transient (Diffusion)	COND, DENS, SPEC

5.2.1 MATERIAL PROPERTY INPUT CARD

**MATE

Status - REQUIRED

Full Keyword - MATERIAL PROPERTY

Function - Signals the beginning of a material property definition.

Input Variables - NONE

Additional Information -

A complete set of material property input must be provided for each material used.

All materials for a problem must be defined before any geometry is specified.

Examples of Use -

1. Define the elastic material properties for a carbon steel.

```
**MATE
  ID STEEL
    EMOD   30.3+6
    POIS   0.30
```

5.2.2 MATERIAL IDENTIFICATION

ID NAME

Status - REQUIRED

Full Keyword - ID

Function - Provides an identifier for a set of material properties related to a given material, thereby allowing later reference to the material property definition.

Input Variables -

NAME (Alphanumeric) - REQUIRED

Additional Information -

The specified name must be unique compared to all other material names included in the problem.

The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

Examples of Use -

1. Define the thermal properties for an aluminum alloy 3003.

```
**MATERIAL
  ID ALUM3003
    COND   25.0
    DENS   0.1
    SPEC   2000.
```

LIBR

Status - OPTIONAL

Full Keyword - LIBRARY

Function - Indicates the material stated by name on the material ID card is contained in a material library.

Input Variables - NONE

Additional Information -

In the present version of the program the material library can only be used for viscoplastic analysis using the Walker's model.

The materials listed below are presently available in the material library. The name listed below should appear on the material ID name.

Available Material Library

ID Name	Material Name	Type of Material
P1100H	IN-100	Precipitation Aged Superalloy (Nickel Based Material)
P1455H	PWA1455	Precipitation Aged Superalloy (Nickel Based Material)
P1038H	HASTELOY-X	Solid Solution Strengthened Superalloy
P1038X	HASTELOY-X	Solid Solution Strengthened Superalloy

NOTE: If HASTELOY-X is desired, then P1038H is recommended. P1038X is similar, however, it uses a different viscoplastic algorithm (integral form).

Examples of Use -

1. Material input for Walker's model

```
**MATE
  ID    P1038H
  LIBR
        INELASTIC
        TIME  5.0   10.0   15.0
        WALKER
$   (end material data set)
```

5.2.3 MASS PARAMETERS

DENS DEN1

Status - (see required material property table)

Full Keyword - DENSITY

Function - Defines the material mass density.

Input Variables -

DEN1 (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Define material properties for a free vibration analysis.

```
**MATE
  ID  STEEL
    EMOD   30.E+6     $      PSI
    POIS   0.30
    DENS   7.324E-4
```

5.2.4 ISOTROPIC ELASTIC PARAMETERS

EMOD EM1

Status - (see required material property table)

Full Keyword - EMODULUS

Function - Defines values of Young's modulus

Input Variables -

EM1 (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Specify a elastic material.

```
**MATERIAL
  ID  MAT1
    EMOD      1.E6
    POIS      0.36
    DENS      0.15
```

POIS   POI

Status - (see required material property table)

Full Keyword - POISSON

Function - Defines the (temperature independent) value of Poisson's ratio.

Input Variables -

POI (Real) - REQUIRED

Allowable values - $1.0 < \text{POI} \leq 0.5$

Additional Information - NONE

Examples of Use -

1. Specify room temperature elastic properties of carbon steel.

```
**MATE
  ID  STEEL
    EMOD  30.E6
    POIS  0.30
```

5.2.5 ISOTROPIC THERMAL PARAMETERS

COND CD1

Status - REQUIRED (for heat conduction analysis)

Full Keyword - CONDUCTIVITY

Function - Defines the isotropic conductivity.

Input Variables -

CD1 (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Specify thermal properties of aluminum for a steady-state heat conduction.

```
**MATE
  ID  ALUM
    CONDUCTIVITY  25.0
```

SPEC SP1

Status - REQUIRED (for heat conduction)

Full Keyword - SPECIFIC

Function - Defines the specific heat.

Input Variables -

SP1 (Real) - REQUIRED

Additional Information -

The user must be careful in selecting appropriate units for specific heat. The CONDuctivity divided by the product of DENSity times SPECific equals the diffusivity. The diffusivity must have units of (Length**2)/time.

Examples of Use -

1. Material model for transient heat conduction.

```
**MATE
  ID  STEEL
    COND   5.8     $      IN.-LB./(SEC.IN.F)
    DENS   0.283   $      LB/(IN3)
    SPEC   1000.   $      IN.-LB./(LB.F)
```

5.2.6 TEMPERATURE-DEPENDENT ISOTROPIC THERMOELASTIC PARAMETERS

```
TEMP      TEM1 TEM2 ...  TEMN
```

Status - (see required material property table)

Full Keyword - TEMPERATURES

Function - Provides the temperature values at which elastic material properties will be defined.

Input Variables -

TEM1 (Real) - REQUIRED

TEM2 ... TEMN (Real) - OPTIONAL

Additional Information -

At least one temperature value must be input. If only one value is input then the properties will be treated as temperature independent.

The temperature values must be specified in ascending order.

This input may be continued on more than one card. Each card must begin with the keyword TEMP.

A maximum of 21 temperature values may be specified for the material.

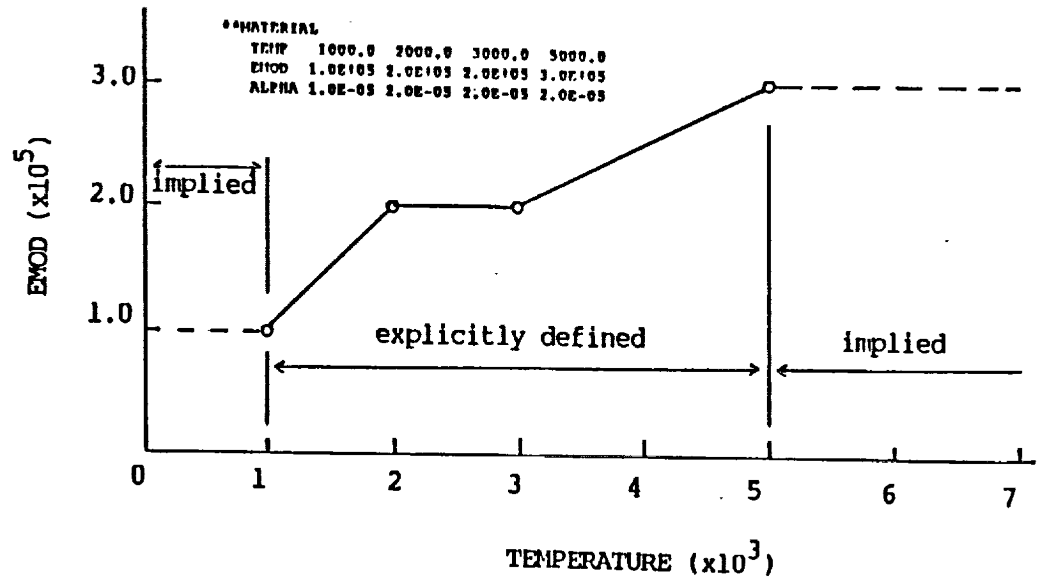
It is important to note that for anisotropy, no more than one value of temperature, viz, TEM1 can be given, i.e., the elastic anisotropic material properties are essentially kept as temperature independent in this version of the code.

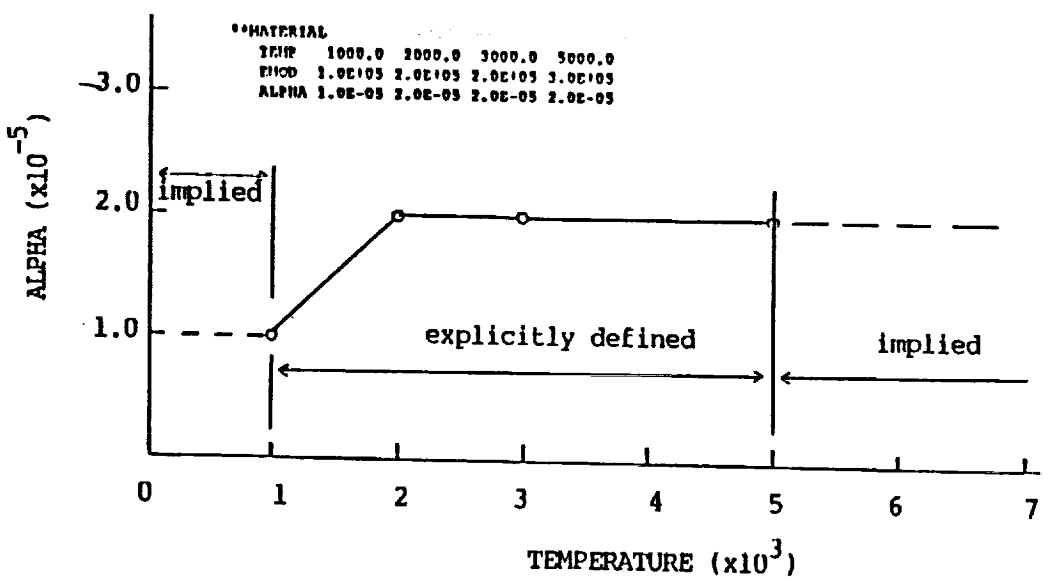
Examples of Use -

1. Define a material with temperature varying Young's modulus.

```
**MATE
  ID  MATOIT
    TEMP  50.0  300.0  500.0
    EMOD  30.3+6   29E+6   27.5E+6
    POIS  0.3
```


C-2





Figures for **MATE: TEMP, EMOD and ALPHA cards

Definition for temperature dependent elastic modulus and thermal coefficient of expansion

EMOD EM1 EM2 ... EMN

Status - (see required material property table)

Full Keyword - EMODULUS

Function - Defines values of Young's modulus at the temperature values specified on the TEMP card(s).

Input Variables -

EM1 (Real) - REQUIRED

EM2 ... EMN (Real) - OPTIONAL

Additional Information -

At least one value must be input.

This input may be continued on more than one card. Each card must begin with the keyword EMOD.

Examples of Use -

1. Specify a thermally-dependent elastic material.

```
**MATERIAL
 ID  MAT1
   TEMP    0.0      200.0     300.0     350.0     400.0
   TEMP    450.0    500.0
   EMOD    1.E6     0.95E6    0.9E6     0.83E6    0.7E6
   EMOD    0.6E6    0.35E6
   POIS    0.36
   DENS    0.15
```

ALPH AL1 AL2 ... ALN

Status - REQUIRED (for thermal stress analysis)

Full Keyword - ALPHA

Function - Defines the values of co-efficient of thermal expansion at the temperature values specified on TEMP cards.

Input Variables -

AL1 (Real) - REQUIRED

AL2 ... ALN (Real) - OPTIONAL

Additional Information -

At least one value must be input.

This input may be continued on more than one card. Each card must begin with the keyword ALPH.

Examples of Use -

1. Define a thermoelastic material model.

```
**MATE
  ID ALUM
    TEMP    0.0     200.0     300.0
    EMOD    10.E3   9.8E3     8.E3
    ALPH    13.E-6  E-6       E-6
    POIS    0.33
```

5.2.7 ANISOTROPIC ELASTIC PARAMETERS

ANIS ITYPE

Status - OPTIONAL

Full Keyword - ANISOTROPY

Function - This card identifies the material as (macroscopic, homogeneous) anisotropic and defines the type of anisotropy.

Input Variables -

ITYPE (Alphanumeric) - REQUIRED

Allowable values are GENE, ORTH, TRAN and CUBI

GENE - General anisotropy. In 3D general anisotropy requires 21 independent constants.

ORTH - Orthotropy or Orthogonal anisotropy (three planes of elastic symmetry). In 3D orthotropy requires 9 independent constants.

TRAN - Traverse isotropy or Cross isotropy or Plane of isotropy. (Directionally solidified material.) In 3D this requires 5 independent constants.

CUBI - Cubic crystal anisotropy. This requires 3 independent constants in 3D.

Additional Information -

Note that for complete isotropy or insignificant anisotropy, the anisotropic elastic material properties should not be used, rather an isotropic elastic analysis should be opted for.

Anisotropy is not available in axisymmetric analysis. The anisotropic material properties are input (as defined on the subsequent pages) for an arbitrary material coordinate system. The material coordinate system can then be rotated to a particular orientation (using the ORIE card) with respect to the global coordinate system which is used to define the geometry of the body.

If geometric and loading symmetry (the SYMM card in **CASE input) is being used in an anisotropic analysis, then the material (in its final orientation) must also be symmetric with respect to the imposed symmetry. This is the users responsibility. Moreover, the SYMM card should not be used with general anisotropy.

Examples of Use -

1. Define option for anisotropic elastic analysis together with the type of anisotropy.

```
**MATE
  ID  MAT1
  TEMP  70.0
  DENSITY  10.0
  ANIS  ORTHO
  TECH  1.31E+5   0.13E+5   0.13E+5   0.038   0.038   0.492   0.064E+5
  TECH  0.032E+5   0.032E+5
```

STIF S11 ... SNN

Status - OPTIONAL (if ANIS is input, either STIF, COMP or TECH must be input)

Full Keyword - STIFFNESSES

Function - Defines the material stiffness for an anisotropic material.

Input Variables -

S11 ... SNN (Real) - REQUIRED in the order defined below

GENE S11, S12, S13, S14, S15, S16, S22, S23, S24, S25, S26
S33, S34, S35, S36, S44, S45, S46, S55, S56, S66

OTHO S11, S12, S13, S22, S23, S33, S44, S55, S66

TRAN S11, S13, S33, S44, S55

CUBI S11, S12, S44

Additional Information -

The relevant stiffnesses appear in the stress-strain relationship as shown for the following cases.

GENE - General Anisotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} & S_{13} & S_{14} & S_{15} & S_{16} \\ & S_{22} & S_{23} & S_{24} & S_{25} & S_{26} \\ & & S_{33} & S_{34} & S_{35} & S_{36} \\ & & & S_{44} & S_{45} & S_{46} \\ & Symm. & & & S_{55} & S_{56} \\ & & & & & S_{66} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

OTHO -Othotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} & S_{13} & 0 & 0 & 0 \\ & S_{22} & S_{23} & 0 & 0 & 0 \\ & & S_{33} & 0 & 0 & 0 \\ & & & S_{44} & 0 & 0 \\ & Symm. & & & S_{55} & 0 \\ & & & & & S_{66} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

TRAN - Transverse Isotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & (S_{11}-2S_{44}) & S_{13} & 0 & 0 & 0 \\ & S_{11} & S_{13} & 0 & 0 & 0 \\ & & S_{33} & 0 & 0 & 0 \\ & & & S_{44} & 0 & 0 \\ & Symm. & & & S_{55} & 0 \\ & & & & & S_{55} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

CUBI - Cubic Crystal Anisotropy

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix} = \begin{pmatrix} S_{11} & S_{12} & S_{12} & 0 & 0 & 0 \\ & S_{11} & S_{12} & 0 & 0 & 0 \\ & & S_{11} & 0 & 0 & 0 \\ & & & S_{44} & 0 & 0 \\ & Symm. & & & S_{44} & 0 \\ & & & & & S_{44} \end{pmatrix} \begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$

IMPORTANT NOTES:

- **The stress-strain relationships defined above are written in terms of engineering strains, not tensor strains.**
- The transversely isotropic material must be input using the convention that the axis of isotropy is the z-axis. The actual orientation of the material for this and other types of anisotropy is set using the ORIE card.
- The material stiffnesses appearing on the STIF card may be split and put onto two or more cards with each new card beginning with the keyword STIF. The order of coefficients, however, must be maintained.

Examples of Use -

1. Input stiffness coefficients for a 3D transverse isotropic medium relative to the principal material axes.

```
**MATE
  ID  MAT1
  TEMP  70.0
  DENSITY  1.0
  ANIS TRAN
  STIFF  2.1601E+7   0.9691E+6   2.0832E+7   0.65E+6   1.0279E+6
```

```
COMP    C11 ...  CNN
```

Status - OPTIONAL (if ANIS is input, either STIF, COMP or TECH must be input)

Full Keyword - `COMPLIANCES`

Function - Defines the material compliances for an anisotropic material.

Input Variables -

C11 ... CNN (Real) - REQUIRED in the order defined below

GENE C11, C12, C13, C14, C15, C16, C22, C23, C24, C25, C26
C33, C34, C35, C36, C44, C45, C46, C55, C56, C66

OTHO C11, C12, C13, C22, C23, C33, C44, C55, C66

TRAN C11, C13, C33, C44, C55

CUBI C11, C12, C44

Additional Information -

The relevant compliances appear in the stress-strain relationship as shown for the following cases.

GENE - General Anisotropy

$$\begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ & & C_{33} & C_{34} & C_{35} & C_{36} \\ & & & C_{44} & C_{45} & C_{46} \\ & Symm. & & & C_{55} & C_{56} \\ & & & & & C_{66} \end{pmatrix} \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

OTHO -Othotropy

$$\begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ & C_{22} & C_{23} & 0 & 0 & 0 \\ & & C_{33} & 0 & 0 & 0 \\ & & & C_{44} & 0 & 0 \\ & Symm. & & & C_{55} & 0 \\ & & & & & C_{66} \end{pmatrix} \begin{pmatrix} \sigma xx \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

TRAN - Transverse Isotropy

$$\begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & (C_{11} - C_{44}/2) & C_{13} & 0 & 0 & 0 \\ & C_{11} & C_{13} & 0 & 0 & 0 \\ & & C_{33} & 0 & 0 & 0 \\ & & & C_{44} & 0 & 0 \\ & Symm. & & & C_{55} & 0 \\ & & & & & C_{55} \end{pmatrix} \begin{pmatrix} \sigma xx \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

CUBI - Cubic Crystal Anisotropy

$$\begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{12} & 0 & 0 & 0 \\ & C_{11} & C_{12} & 0 & 0 & 0 \\ & & C_{11} & 0 & 0 & 0 \\ & & & C_{44} & 0 & 0 \\ & Symm. & & & C_{44} & 0 \\ & & & & & C_{44} \end{pmatrix} \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

IMPORTANT NOTES:

- **The stress-strain relationships defined above are written in terms of engineering strains, not tensor strains.**
- The transversely isotropic material must be input using the convention that the axis of isotropy is the z-axis. The actual orientation of the material for this and other types of anisotropy use the ORIE card.
- The material compliances appearing on the COMP card may be split and put onto two or more cards with each new card beginning with the keyword COMP. The order of coefficients, however, must be maintained.

Examples of Use -

1. Input compliance coefficients for a 3D transverse isotropic medium relative to the principal material axis.

```
**MATE
  ID  MAT1
  TEMP  70.0
  DENSITY  1.0
  ANIS TRAN
  COMP  4.7619E-7   -1.4762E-8   6.4516E-7   1.5385E-8   7.2750E-8
```

TECH E1 E2 E3 ν12 ν13 ν23 G12 G13 G23

Status - OPTIONAL (if ANIS is input, either STIF, COMP, or TECH must be input)

Full Keyword - TECHNICAL

Function - Inputs pertinent technical (or engineering) constants for an **orthotropic** material.

Input Variables -

E1,E2,E3,ν12,ν13,ν23, G12,G13,G23 (Real) - REQUIRED

Additional Information -

The TECH card can only be used for orthotropic analysis.

For the present case of an orthotropic material the technical constants may be described most conveniently by the following strain-stress relationship:

$$\begin{pmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{pmatrix} = \begin{pmatrix} 1/E_1 & -\nu_{12}/E_1 & -\nu_{13}/E_1 & 0 & 0 & 0 \\ & 1/E_2 & -\nu_{23}/E_2 & 0 & 0 & 0 \\ & & 1/E_3 & 0 & 0 & 0 \\ & & & 1/G_{12} & 0 & 0 \\ & & & & 1/G_{13} & 0 \\ & Symm. & & & & 1/G_{23} \end{pmatrix} \begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{pmatrix}$$

where $E_1\nu_{21} = E_2\nu_{12}, E_2\nu_{32} = E_3\nu_{23}, E_3\nu_{13} = E_1\nu_{31}$

IMPORTANT NOTES:

- **The stress-strain relationships defined above are written in terms of engineering strains, not tensor strains.**
- The material parameter appearing on the TECH card may be split and put onto two or more cards with each new card beginning with the keyword TECH. The order of coefficients, however, must be maintained.

Examples of Use -

1. Input technical constants (also called 'engineering constants') relevant to a 3D analysis for an orthotropic medium relative to the principal material axes.

```
**MATE
  ID  MAT1
  TEMP  70.0
  DENSITY  1.0
  ANIS  ORTH
  TECH  21.0E+6    1.55E+6    1.55E+6
  TECH  0.31       0.31       0.49
  TECH  0.65E+6    0.5E+6     0.5E+6
```

```
ORIE      A1 ...  BN
```

Status - OPTIONAL

Full Keyword - `ORIENTATION`

Function - Defines the orientation of material axes in an anisotropic analysis with respect to the global axes of the problem. The geometry of the body is defined with respect to the global axes.

Input Variables -

A1,A2,A3,B1,B2,B3 (Real) - REQUIRED

where

A1, A2, A3 are the cartesian components of a vector parallel to the material x-axis in the global (geometric) coordinate system, and

B1, B2, B3 are the cartesian components of a vector parallel to the material y-axis in the global (geometric) coordinate system.

Additional Information -

This card can be omitted if the material axes are aligned with the geometric axes.

The STIFF, COMP, TECH and ALPH cards when present must precede the ORIE card.

If geometric and loading symmetry (the SYMM card) is being used in an anisotropic analysis then the material (after rotation) must be symmetric with respect to the imposed symmetry. This is the user's responsibility. Moreover, the SYMM card should not be used with general anisotropy.

The x-axis and the y-axis defined on the ORIE card must be at right angles. The third (z) axis will be calculated by BEST3D.

Examples of Use -

1. For 3D transverse isotropy, provide the directional cosines for material axes at an oblique rotation.

```
**MATE
  ID  MAT1
  TEMP  70.0
  DENSITY  1.0
  ANIS  TRAN
  COMP  4.76E-7  -1.47E-8  6.46E-7  1.538E-7  7.27E-7
  ORIE  0.7071    0.7071   0.7071  -0.7071    0.7071   0.7071
```

5.2.8 ISOTROPIC VISCOUS PARAMETERS

DAMP DAMPR

Status - REQUIRED (for steady-state forced vibration or Laplace domain transient dynamic analysis)

Full Keyword - DAMPING

Function - Defines the viscous damping co-efficient for the material.

Input Variables -

DAMPR (Real) - REQUIRED

Additional Information - NONE

Examples of Use -

1. Define the viscous damping coefficient for material MAT1

```
**MATERIAL
  ID MAT1
  TEMP 70.
  EMOD 2.6666
  POIS 0.3333
  DENS 1.0
  DAMP 0.05
```

5.2.9 ADDITIONAL ELASTO-PLASTIC AND VISCO-PLASTIC PARAMETERS

INEL

Status - REQUIRED (for plastic analysis)

Full Keyword - `INELASTIC`

Function - Identifies the fact that an inelastic material model will be defined.

Input Variables - NONE

Additional Information -

If the INEL card is used during a material property definition, the program will then expect one of four models to be chosen. These models are specified using the VON, TWO or WALK keywords. Only one of these cards may be used in a single material property input data set. Input for the nonlinear models is described below.

NOTE: In present version of the program, all non-linear materials used in an analysis must be based on the same model.

Examples of Use -

1. Request an inelastic analysis using the Von Mises Material Model.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.
  POIS 0.2
  INELASTIC
    VON MISES
```

`TIME    I1 I2 ...  IN`

Status - OPTIONAL

Full Keyword - `TIME`

Function - Defines time steps at which the accelerated nonlinear solution algorithm should not be used.

Input Variables -

I1 (Integer) - REQUIRED

I2 ... IN (Integer) - OPTIONAL

Additional Information -

In general it is recommended that the TIME card be used to turn off the accelerated algorithm whenever the loading is reversed. The accelerated algorithm attempts to extrapolate initial stress based on recent history and extrapolate in the wrong direction at times when loading reverses.

A maximum of 20 time steps may be input.

Examples of Use -

1. Specify a reversal of loading takes place at two time step numbers during cyclic loading of a material.

```
**MATERIAL
    ID MAT1
    TEMP 70.0
    EMOD 23.8E+03
    POIS 0.3
  INELASTIC
    TWO SURFACE
    YIELD 100.0 400.0
    HARD 35.7E+03 11.9E+03
    TIME 7 17
```

5.2.9.1	VON MISES MODEL

VON

Status - OPTIONAL

Full Keyword - VON-MISES-MODEL

Function - Selects the use of the inelastic Von Mises Model in which the stress-strain curve is defined as a set of equivalent stress-equivalent plastic strain pairs, with linear interpolation between points.

Input Variables - NONE

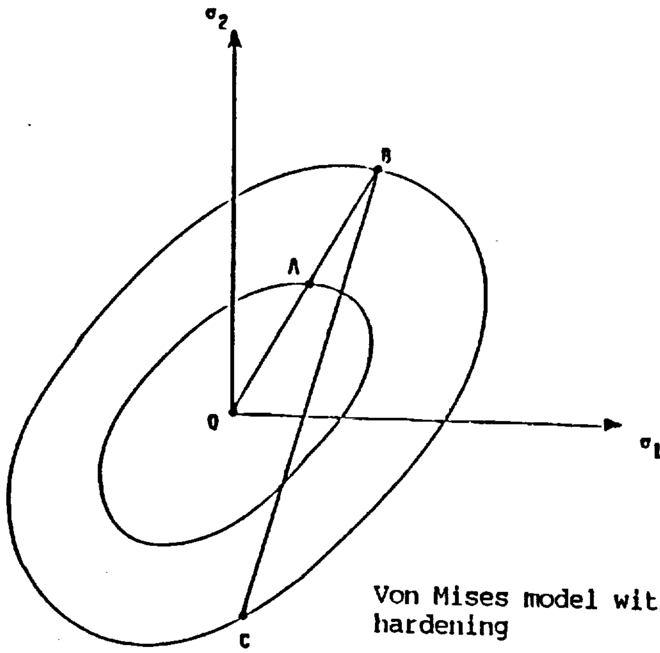
Additional Information -

The input for this model is extremely easy to generate if a monotonic stress-strain curve is available for the material. The model is intended primarily for the analysis of monotonic loading situations. For applications involving cyclic loading and reverse yielding use of the two-surface model (keyword=TWO) is recommended.

Examples of Use -

1. Request the use of the Von Mises Model for an inelastic analysis.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.0
  POIS 0.2
  INELASTIC
    VON MISES
```



Von Mises model with variable strain hardening

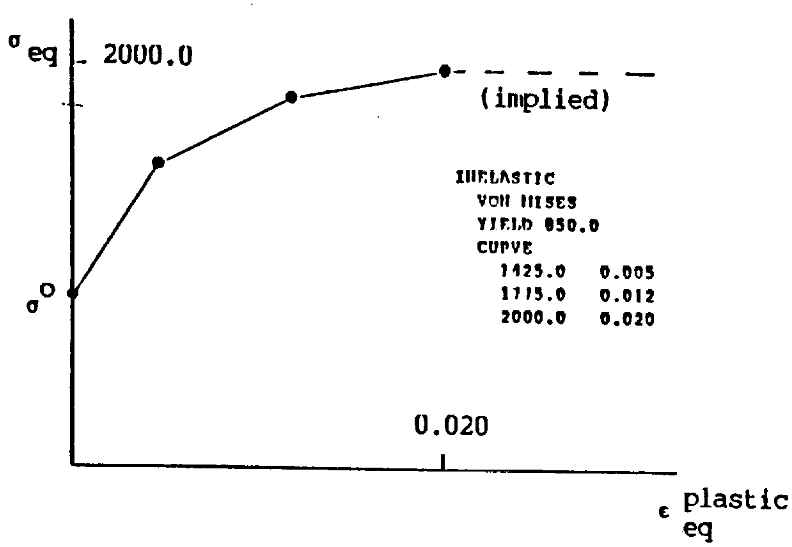


Figure for **MATE: VON (YIELD and CURVE) cards

VON MISES equivalent stress/equivalent plastic strain hardening curve

YIEL Y1

Status - REQUIRED (if VON is input)

Full Keyword - YIELD

Function - Defines the proportional limit for the isotropic inelastic material models.

Input Variables -

Y1 (Real) - REQUIRED

Inner proportional limit.

Additional Information -

Y1 is the normal proportional limit, at which the transition away from purely elastic behavior occurs.

Examples of Use -

1. Specify the yield stress value at which the material ceases to behave in an elastic manner.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.0
  POIS 0.2
  INELASTIC
    VON MISES
    YIELD 24.3
```

CURV

Status - REQUIRED (with VON only)

Full Keyword - CURVE

Function - Identifies the beginning of the input defining the stress-strain curve for the isotropic model.

Input Variables - NONE

Additional Information -

This card is immediately followed by input cards defining the points on the equivalent stress - equivalent plastic strain curve. These data cards are described immediately below.

(NONE) SIG1 EP1

Status - REQUIRED (if CURV is input)

Full Keyword - NO KEYWORD REQUIRED

Function - To define a single point on the material stress strain curve.

Input Variables -

SIG1 (Real) - REQUIRED

Stress value

EP1 (Real) - REQUIRED

Equivalent plastic strain value

Additional Information -

This card type is input once for each point used to describe the stress-strain curve. The stress values must be monotonically increasing. The point (yield stress,0) is inserted automatically by the program and must not be input by the user.

In order to define a perfectly plastic material the user should input a single point at which the stress is the yield stress and the equivalent plastic strain has any positive value.

A maximum of Nine points (excluding the initial point can be input by the user.

Linear interpolation is used between points. Perfectly plastic material is assumed after the last point.

Examples of Use -

1. Define the equivalent stress-equivalent plastic strain curve for a Von Mises material.

```
**MATERIAL
  ID MAT1
  TEMP 70.0
  EMOD 7000.0
  POIS 0.2
  INELASTIC
    VON MISES
    YIELD 100.
    CURVE
      150. 0.25
      180. 0.5
      190. 0.75
      200. 1.0
```

5.2.9.2	TWO-SURFACE MODEL

TWO

Status - OPTIONAL

Full Keyword - `TWO SURFACE`

Function - Selects use of the two-surface inelastic material model.

Input Variables - NONE

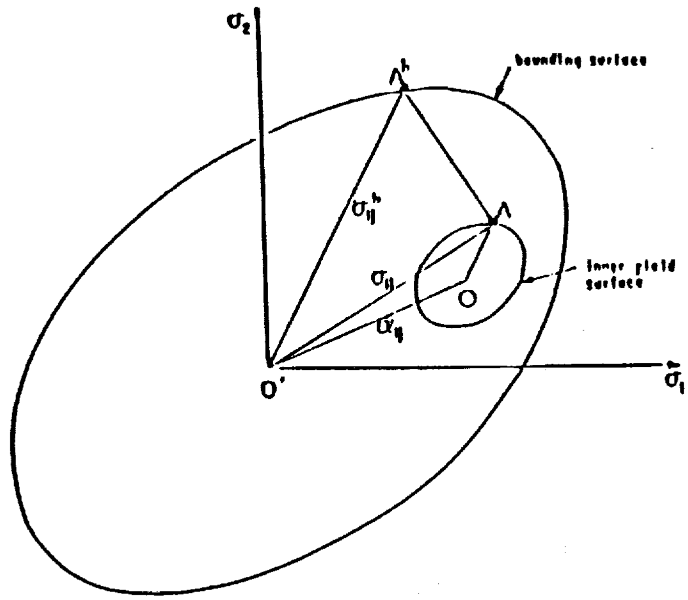
Additional Information -

This model is recommended for either monotonic or cyclic loading situations.

Examples of Use -

1. Select the use of the two-surface inelastic material model.

```
**MATERIAL
    ID MAT1
    TEMP 70.0
    EMOD 23.8E+03
    POIS 0.3
  INELASTIC
    TWO SURFACE
```



Two-surface plasticity model

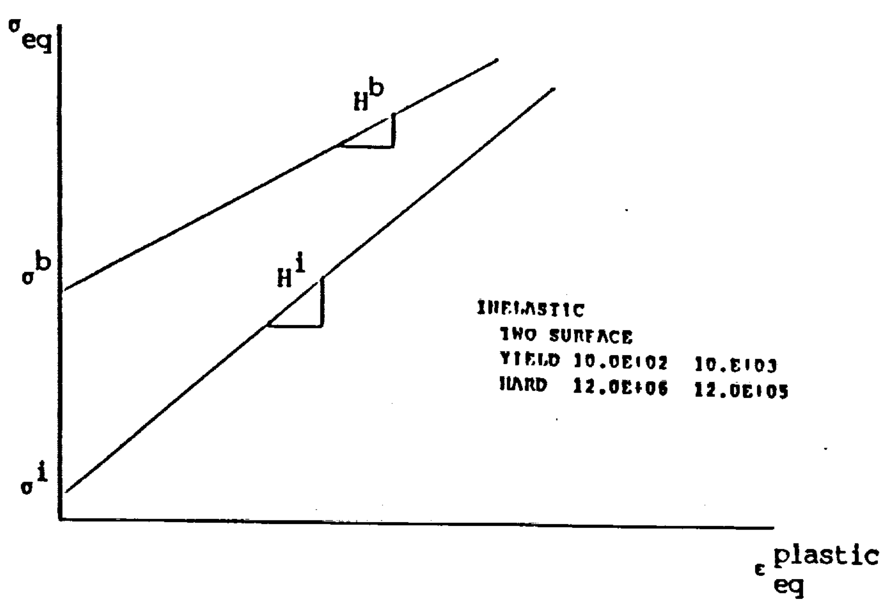


Figure for **MATE: TWO (YIELD and HARD) cards

TWO SURFACE equivalent stress/equivalent plastic strain hardening curve

YIEL Y1 Y2

Status - REQUIRED (if TWO is input)

Full Keyword - YIELD

Function - Defines the proportional limits for the two-surface inelastic material models.

Input Variables -

Y1 (Real) - REQUIRED
Inner proportional limit.

Y2 (Real) - REQUIRED
Outer proportional limit.

Additional Information -

Y1 is the normal proportional limit, at which the transition away from purely elastic behavior occurs. Y2 is the stress value at which equivalent plastic strain can be reasonably approximated as a linear function of stress. Alternatively, Y2 can be taken as any stress larger than the highest stress anticipated to occur in the solution.

Examples of Use -

1. Specify the values of inner and outer yield stresses for the two-surface material model.

```
**MATE
    ID MAT1
    TEMP 70.0
    EMOD 23.8E+03
    POIS 0.3
  INELASTIC
    TWO SURFACE
    YIELD 100.0 400.0
```

HARD HR1 HR2

Status - REQUIRED (with TWO only)

Full Keyword - **HARD**

Function - Defines the hardening parameters at the inner and outer proportional limits for the two surface model.

Input Variables -

HR1 (Real) - REQUIRED

HR1 = at Y1

HR2 (Real) - REQUIRED

HR2 = at Y2

Additional Information -

Based on the values of the inner and outer hardening and proportional limits, **BEST3D** calculates a Ramberg-Osgood stress-strain law to be used in the region between the two proportional limits.

Examples of Use -

1. Specify the hardening parameter values for the inner and outer surfaces for a two-surface material model.

```
**MATE
    ID MAT1
    TEMP 70.0
    EMOD 23.8E+03
    POIS 0.3
  INELASTIC
    TWO SURFACE
    YIELD 100.0 400.0
    HARD 35.7E+03 11.9E+03
```

5.2.9.3	WALKER'S MODEL

WALK

Status - OPTIONAL

Full Keyword - `WALKER`

Function - Selects the viscoplastic material model developed by Kevin Walker.

Input Variables - NONE

Additional Information -

The Walker's model must be used in conjunction with one of the predefined materials contained in the material library. The name of the material from the material library must be stated by name of the material ID card and the LIBR card must be included. No other material parameter values are required or allowed for this model.

References for the Walker viscoplastic model:

1. K.P. Walker, "Research and Development Program for Nonlinear Structural Modeling with Advanced Time-Temperature Dependent Constitutive Relationships," NASA CR-165533, November 1981.
2. B.N. Cassenti and R.L. Thompson, "Material Response Predictions for Hot Section Gas Turbine Engine Components," AIAA-83-2020, presented at the AIAA/SAE/ASME 19th Joint Propulsion Conference, Seattle, Washington, June 27-29, 1983.

Examples of Use - Material input for Walker's model.

```
**MATE
  ID  P1038H
  LIBR
    INEL
    TIME  5.0   10.0   15.0
    WALKER
$ (end of material input)
```

5.3 DEFINITION OF GEOMETRY

In the current version of **BEST3D**, surface geometry is defined using six, seven, eight and nine noded parametric surface patches. These patches can be defined to have either linear or quadratic variation of the primary field variables. Additionally, holes on the interior of a 3-D body can be defined via two and three noded line elements. The three-noded hole elements can have either linear or quadratic functional variation in the longitudinal direction. An entire model may be assembled from several generic modelling regions (GMR). Each generic modelling region is defined in a single block of input introduced with a **GMR card.

The information provided in a single GMR input block consists of six main types:

1 - Region identification
2 - Nodal point definition
3 - Surface connectivity definition
4 - Hole or Insert definition (if desired)
5 - Volume cell connectivity or global shape function definition (if required)
6 - Sampling point definition (if desired)

(Note: Holes, Inserts, Volume cells and Global shape functions are not presently available for all Analysis Types.)

A list of keywords recognized in the GMR input are given below and a detailed description follows.

SECTION	KEYWORD	PURPOSE
5.3.1 Geometry Input Card		
	**GMR	start of generic modelling region input
5.3.2 Region Identification		
	ID	region ID
	MATE	material property(set) for region
	TREF	reference temperature of region
	TINI	initial temperature of the region
	EXTE	region is an infinite body
	HALF	region is a half-space

SECTION	KEYWORD	PURPOSE
5.3.3 Nodal Point Definition		
	POIN	nodal points for boundary and volume discretization
	(Coordinates)	
5.3.4 Surface Element Definition		
	SURF	beginnining of surface discretization
	TYPE LINE	linear surface variation of field quantities
	TYPE QUAD	quadratic surface variation of field quantities
	ELEM	element list
	(element connectivity)	
	TRAN	surface generation by translation
	REFE	axis of rotation for surface generation
	DIRE	direction of axis of rotation
	ROTA	angle of rotation for surface generation
	NORM	defines outer normal of surface
5.3.5 Enclosing Element Definition		
	ENCL	enclosing element list
	(enclosing element connectivity)	
5.3.6 Hole Element Definition		
	HOLE	start of hole definition
	POIN	nodal points for hole discretization
	(coordinates)	
	TYPE LINE	linear variation of field quantities for 3-noded holes
	TYPE QUAD	quadratic variation of field quantities for 3-noded holes
	ELEM	hole element list
	(element connectivity and radius of hole)	

SECTION	KEYWORD	PURPOSE
5.3.7 Insert Element Definition		
	INSE	beginning of Insert definition
	POIN	nodal points for insert discretization
	(coordinates)	
	TYPE LINE	linear variation of field quantities of Insert
	TYPE QUAD	quadratic variation of field quantities of Insert
	ELEM	start of connectivity for each Insert
	(element connectivity and radius of insert)	
5.3.8 Volume Cell Definition		
	VOLU	beginning of volume discretization
	TYPE LINE	linear variation of cell quantities
	TYPE QUAD	quadratic variation of cell quantities
	CELL	volume cell definition
	(cell connectivity)	
	FULL	region completely filled with cells
5.3.9 Global Shape Function Definition		
	GLOB	global shape function in GMR
	NODE	global shape function node list
	(node list)	
5.3.10 Sampling Points		
	SAMP	start of definition of sampling points
	(coordinates)	

5.3.1 GEOMETRY INPUT CARD

****GMR**

Status - REQUIRED

Full Keyword - `GMREGION`

Function - This card signals the beginning of the definition of a generic modelling region.

Input Variables - NONE

Additional Information -

At least one GMR must be defined for an analysis. If more than one GMR is defined, then the input for each is initiated with a **GMR card.

GMR definitions must all precede all Interface, Boundary Condition set, and Body Force Definitions. Each GMR must be a closed region of three dimensional space. However, under the following two circumstances, the region may be open :

1 - In planar symmetry problems, the body may be sliced into symmetric parts and only one of these parts requires discretization. The interior section exposed by the plane cutting the body does not represent a boundary, and therefore it does not require discretization.

2 - In GMRs with boundaries extending to infinity, a GMR may have open boundaries. However, this must be indicated through the use of the EXTE or HALF cards or by enclosing the open boundary with Enclosing elements (see the ENCL card). Note : One of the above three devices MUST be used in an infinite region. Infinite elements may optionally be used but it cannot replace the use of the EXTE, HALF or ENCL cards.

A GMR may have multiple internal boundaries in addition to a single external boundary.

Examples of Use -

```
**GMR
  ID  REG1
  MATE  STEEL
  TREF  70.0
  TINI   0.0
  POINT
    1  10.0  0.0  2.0
    2  10.0  1.0  2.0
              .
              .
```

5.3.2 REGION IDENTIFICATION

ID NAME

Status - REQUIRED

Full Keyword - ID

Function - This card provides the identifier for the GMR.

Input Variables -

NAME (Alphanumeric) - REQUIRED

Additional Information -

The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

The name provided on this card is used to reference the GMR in other portions of the input as well as in the **BEST3D** output file.

The NAME must be unique compared to all the other GMR names defined in the problem.

Examples of Use -

```
**GMR
  ID REG1
  MATE STEEL
```

MATE NAME

Status - REQUIRED

Full Keyword - MATE

Function - This card identifies the material property set for the GMR.

Input Variables -

NAME (Alphanumeric) - REQUIRED

Additional Information -

The material name reference must have been previously defined in the material property input (identified as NAME on the ID card in **MATE input).

Examples of Use -

```
**GMR
  ID GMR1
  MATE STEEL
```

TREF TEMP

Status - OPTIONAL

Full Keyword - TREFERENCE

Function - This card defines the reference temperature at which the material properties will be evaluated for use in integration of this GMR.

Input Variables -

TEMP (Real) - REQUIRED

Additional Information -

If temperature dependent material properties were input in **MATE, the properties for the GMR will be calculated, based on the temperature specified on this card, using linear interpolation.

For problems in which temperature changes in time and/or space, it is recommended that the reference temperature be chosen as the (time/volume weighted) average temperature over the GMR.

If this card is not input then a reference temperature of zero is assumed.

Examples of Use -

1. Specify the reference temperature at which the material properties are evaluated.

```
**GMR
  ID REG1
  MATE MAT1
  TREF 70.0
```

TINI TEMP

Status - OPTIONAL (used in temperature dependent problems)

Full Keyword - `TINITIAL`

Function - This card defines the initial temperature (i.e. the datum temperature or the zero stress-strain state) of the region at the beginning of a temperature dependent problem.

Input Variables -

TEMP (Real) - REQUIRED

Additional Information -

If this card is not input, the initial temperature is assumed as zero.

Examples of Use -

1. Specify the initial temperature of the region REG1.

```
**GMR
  ID REG1
  MATE MAT1
  TREF 70.0
  TINI  0.0
```

EXTE

Status - OPTIONAL

Full Keyword - EXTERIOR

Function - This card identifies that the present GMR is a part of a infinite region.

Input Variables - NONE

Additional Information -

The entire outer boundary of the GMR must extend to infinity.

Infinite elements should not be used in the GMR.

In an analysis of a problem of a body of infinite extent, it is not neccessary to fix the boundary of the body for the sole purpose of preventing rigid body motions. Basically, the mathematics of the problem assumes zero displacement at infinity.

When the entire outer boundary of a GMR is at infinity (e.g cavity in an infinite space) the outer boundary can not and should not be modeled. Instead the EXTE card should be inserted in the GMR input to indicate this fact. The purpose of this card is to account for the contributions of the unmodeled infinite boundary in the calculation of the diagonal terms of the **F** matrix (Rigid body translation technique).

An alternative method to account for infinite boundaries is to model the infinite boundary with enclosing elements (see ENCL card). However, this is not recommended in problems when the entire outer boundary extends to infinity, since the use of enclosing elements would be more expensive then using the EXTE card.

Examples of Use -

1. Specify that the region GMR1 is part of an infinite region.

```
**GMR
  ID  GMR1
  MAT MAT1
  TREF 70.0
  EXTERIOR
  POINTS
        1     0.0      212.00     0.0
        2    41.36     207.93     0.0
        .     .          .         .
        .     .          .         .
        .     .          .         .
```

HALF

Status - OPTIONAL

Full Keyword - `HALF`

Function - This card identifies that the present GMR is part of a half-space.

Input Variables - NONE

Additional Information -

In an analysis of a problem of a body of infinite extent, it is not neccessary to fix the boundary of the body for the sole purpose of preventing rigid body motions. Basically, the mathematics of the problem assumes zero displacement at infinity.

In order to use the HALF card, the GMR must be a half space and the half-space must be modeled with regular boundary elements about the area of interest. As the boundary extends to infinity, the boundary elements can be truncated at a reasonable distance away from the area of interest (or infinite elements can be used).

The purpose of this card is to account for the contribution of the unmodeled (semi-infinite) boundary in the calculation of the diagonal terms of the **F** matrix (Rigid body translation technique)

The part of the half-space that is modeled with regular boundary elements may contain irregularities such as a trench, however, the distant elements that extend towards infinity must lie on a plane.

An alternative method to account for the semi-infinite boundary is to model the boundary with enclosing elements (see ENCL card), however, this is not recommended in half-space problems since the use of enclosing elements would be more expensive then using the HALF card.

Examples of Use -

1. Specify that the region GMR1 is a part of a half space.

```
**GMR
  ID  GMR1
  MAT MAT1
  TREF 70
  HALF
  POINTS
        1     0.00     0.00     0.00
        2     0.70     0.70     0.00
        .       .        .        .
        .       .        .        .
```

5.3.3 NODAL POINT DEFINITION

POIN

Status - REQUIRED (for defining the GMR)

Full Keyword - `POINTS`

Function - This card initiates the definition of nodal points for the boundary element and volume cell discretization (or global shape function definition) of the GMR.

Input Variables - NONE

Additional Information -

Under certain conditions **BEST3D** can automatically generate nodal points. This is done by translation and/or rotation of previously defined surfaces. At the beginning of the definition of the first GMR some points must be defined in order to begin the process, although explicit definition of all nodal points in the first GMR is not required. It is possible that GMRs defined subsequent to the first may require no explicit nodal point input. For example, the case when a GMR is obtained by translation and/or rotation of previously defined surfaces.

Nodal points used for hole and insert elements discretization **CANNOT** be defined here. Instead, the nodal points for holes and inserts must be defined under their respective section.

Sampling Points for which results are requested (at any point on or in the body) is input under the Sampling Point section.

(NONE) NNODE X Y Z

Status - REQUIRED

Full Keyword - NO KEYWORD REQUIRED

Function - This card defines the node number and the Cartesian coordinates for a single nodal point.

Input Variables -

NNODE (Integer) - REQUIRED

User node number for the node.

X,Y,Z (Real) - REQUIRED

Cartesian coordinates of the node.

Additional Information -

This card is input once for each point.

User node numbering must be unique.

All node numbers must be less than or equal to 9999.

Nodal coordinates for both surface and volume discretization should be input here. If a node is not referenced in the surface or volume discretization, then it is ignored.

Nodal points used for hole and insert elements **CANNOT** be defined here. Instead, the nodal points for holes and inserts must be defined under their respective section.

Sampling Points for which results are requested (at any point on or in the body) is input under the Sampling Point section.

Examples of Use -

1. Define a set of nodal coordinates in GMR1.

```
**GMR
  ID   GMR1
  MAT MAT1
  TREF 70.0
  HALF
  POINTS
        1     0.000     0.000     0.000
        2     0.700     0.700     0.000
        3     0.700     0.000     0.000
        4     0.000     0.700     0.000
        5     1.000     0.000     0.000
        .     .         .  .
        .     .         .  .
        .     .         .  .
```

5.3.4 SURFACE ELEMENT DEFINITION

SURF NAME REFNAME

Status - REQUIRED (minimum of one per GMR)

Full Keyword - SURFACE

Function - This card initiates the definition of a surface of the current GMR.

Input Variables -

NAME (Alphanumeric) - REQUIRED

The name of the surface being defined.

REFNAME (Alphanumeric) - OPTIONAL

The name of a previously defined surface which will be used to create the current surface (defined as NAME on SURF card in prior GMR input).

Additional Information -

The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

The names assigned to the various surfaces in the problem must be unique.

Two techniques are available to define a surface.

1 - A set of surface elements may be defined using explicitly defined nodes. In this case REFNAME is not input, and the card following the SURF card must be a TYPE card. (The first surface in the first GMR must be defined in this way).

2 - If REFNAME is input, then that (previously defined) surface will be used to define the surface NAME through translation and rotation. A TRAN card and/or a set of cards consisting of a REF, DIR and ROT cards must be included. Translation takes priority over rotation. Any new nodal points required will be automatically generated, and any duplicate nodes will be eliminated. The TYPE variation of the new surface is automatically assumed to be the same as the reference surface (this cannot be changed).

Boundary conditions and/or interface conditions may be applied to the automatically generated surfaces, however, it is the user responsibility to insure that the elements and nodes generated by the program are referenced by the correct numbers. The RESTART WRITE option may be useful in this respect. In any case, boundary conditions that are applied to the entire surface can be input without complications.

The current verions of **BEST3D** allows RENAME to refer to a surface previously defined in the current GMR.

Examples of Use -

1. Define a 3-D quadratic surface named SIDE

```
SURFACE  SIDE
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
 1003  5  6  7  107  207  206  205  105
 1004  7  8  1  105  201  208  207  107
NORMAL 101 +
```

2. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE  SURF1
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFACE SURNEW SURF1
 TRAN   1.0  -1.0   0.0
 REFE   1.0   0.0   0.0
 DIRE   0.0   0.0   1.0
 ROTA   180.0
NORMAL 1001 +
```

TYPE ETYPE

Status - REQUIRED (if REFNAME not input)

Full Keyword - TYPE

Function - This card defines the variation of field quantities over the elements of the current surface.

Input Variables -

ETYPE (Alphanumeric) - REQUIRED

Allowable values are LINE (linear shape function for field variables) or QUAD (quadratic shape function for field variables)

Additional Information -

All of the elements of a single surface must have the same type of variation. Different surfaces of the same GMR may have different variation. When surfaces of different variation type meet, the element sides along the junction are constrained to have linear variation.

A surface may consist of a single element. By contrast a single surface may define the entire boundary of a GMR.

Examples of Use -

1. Specify that the field quantities vary quadratically over the elements of the surface SURF1.

```
SURFACE  SURF1
TYPE  QUAD
ELEMENT
    101   1   2   3   103   203   202   201   101
    102   3   4   5   105   205   204   203   103
     .    .   .   .    .     .     .     .     .
     .    .   .   .    .     .     .     .     .
     .    .   .   .    .     .     .     .     .
     .    .   .   .    .     .     .     .     .
```

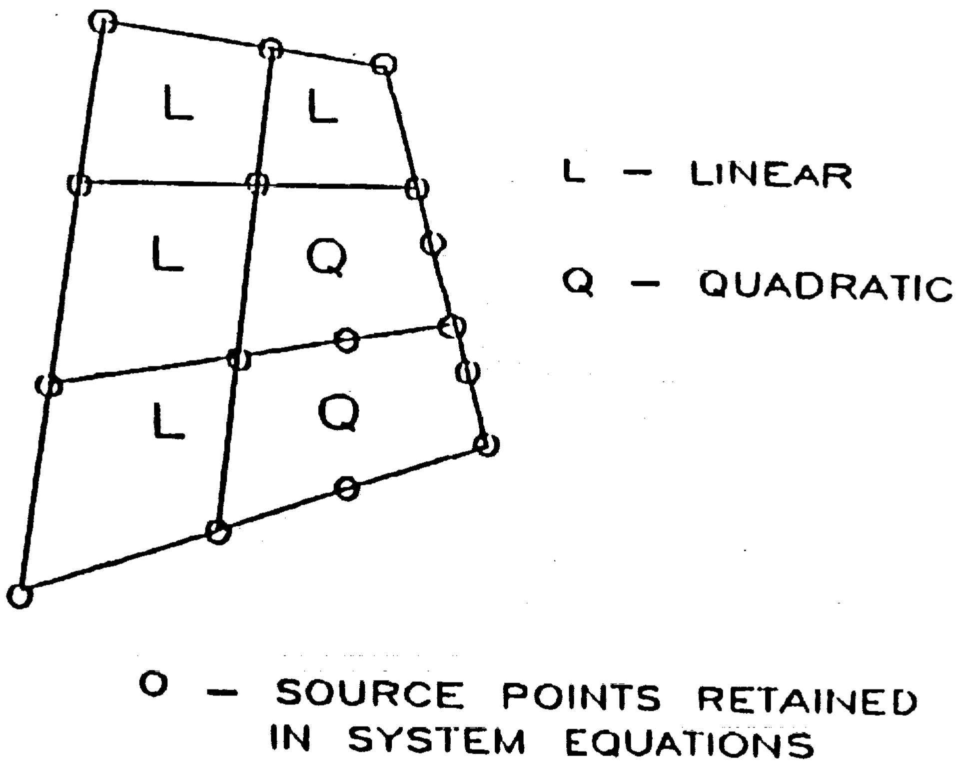


Figure for **GMR: TYPE card
Effect of mixed variation

ELEM

Status - REQUIRED (if REFNAME not input)

Full Keyword - ELEMENTS

Function - Signals the beginning of the connectivity definition for surface elements of the current surface.

Input Variables - NONE

Additional Information - NONE

(NONE) **IFLAG NEL NODE1 ... NODEN NODEREF**

Status - REQUIRED (minimum of one card if TYPE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the connectivity for a single surface element.

Input Variables -

IFLAG (Alphanumeric) - OPTIONAL

IFLAG is omitted except in the case of an infinite element. IFLAG = I identifies an infinite element.

NEL (Integer) - REQUIRED

User element number.

NODE1 ... NODEN (Integer) - REQUIRED

User node numbers of the six, seven, eight or nine node for defining the geometry of the element. Every surface patch must have three, six, seven, eight or nine nodes, regardless of whether TYPE = LINE or QUAD.(The shape functions for geometry is always quadratic)

NODEREF (Integer) - OPTIONAL

User node number for reference node used in infinite element definition. If IFLAG = I and NODEREF is omitted the reference node will be taken to be the origin.

Additional Information -

This card is input once for each element.

The input card need not specify whether a three, six, seven, eight or nine node element is being defined. The starting point in may be any of the corner nodes, however the direction of input around the element is arbitrary, as long as, input is consecutive.

User element numbers must be unique and less than or equal to 9999.

In the definition of an infinite element the first node input must lie on the fixed edge of the element.

All of the nodes referenced in the surface element connectivity must have been defined previously in POINts.

In GMRs of infinite extent, infinite elements may or may not be used. However, whenever a GMR is of infinite extent either the EXTE or HALF card must be used or ENCLosing elements must be defined.

Examples of Use -

1. Specify the connectivity definition for elements of the surface SIDE using four 8-noded quadratic elements.

```
SURFACE  SIDE
TYPE  QUAD
ELEMENT
   101   1   2   3   103   203   202   201   101
   102   3   4   5   105   205   204   203   103
   103   5   6   7   107   207   206   205   105
   104   7   8   1   105   201   208   207   107
NORMAL 101 +
```

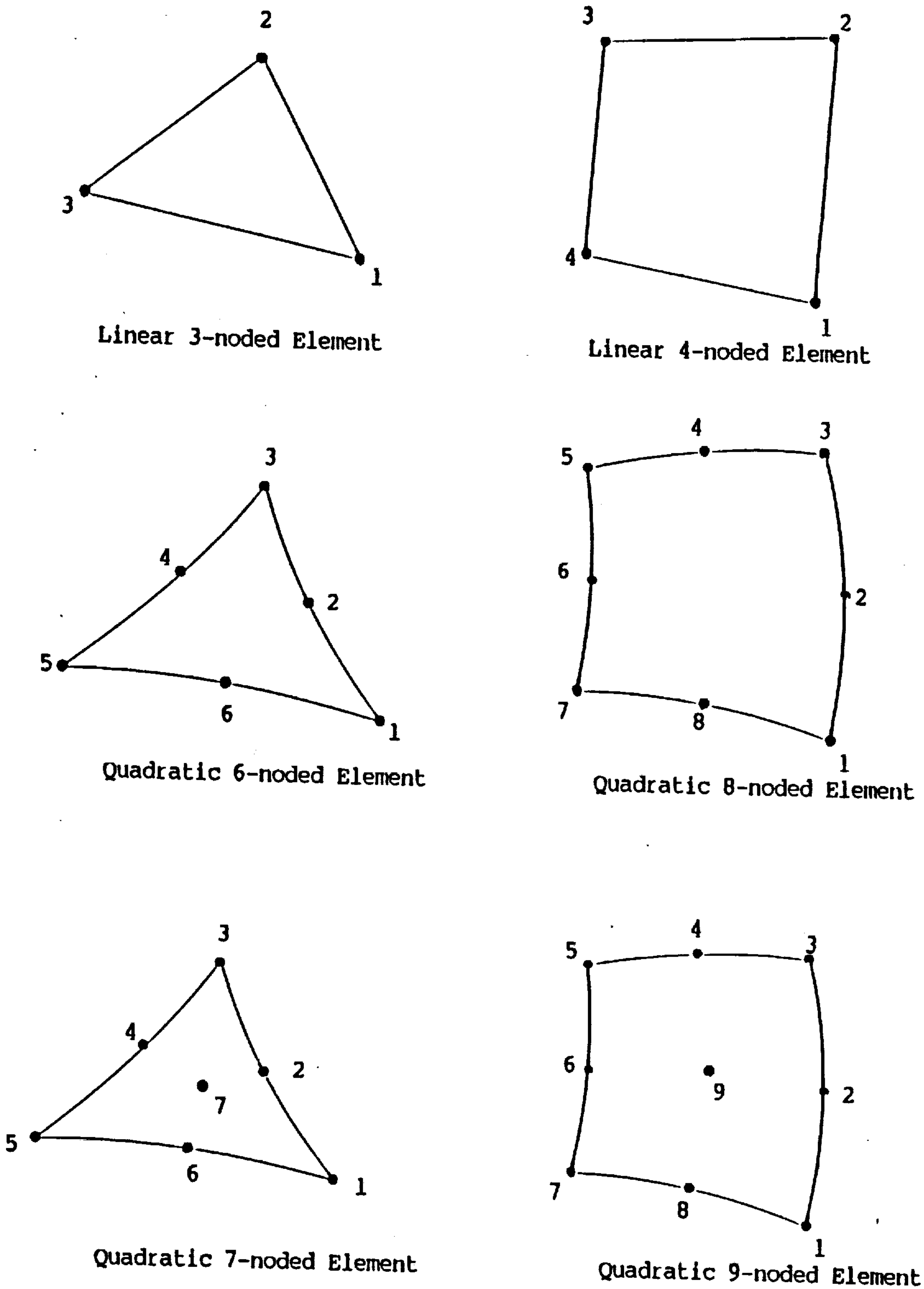
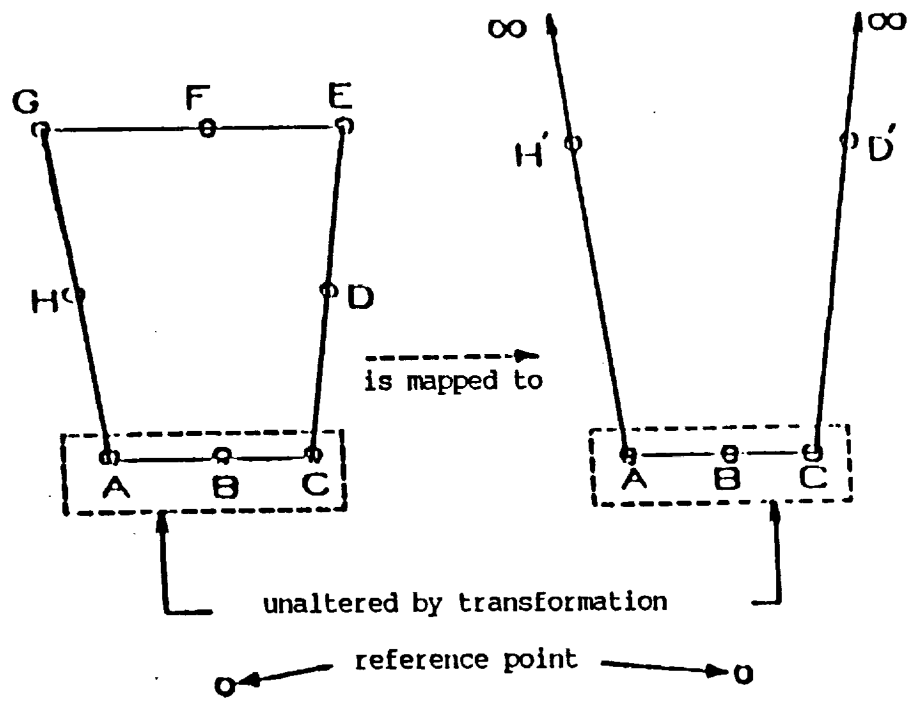


Figure for **GMR: element Connectivity card
Three-dimensional (surface) boundary elements



NODE A OR C MUST BE
THE FIRST NODE INPUT
ON AN INFINITE ELEMENT

Figure for **GMR: (element) Connectivity card
Definition of infinite elements

TRAN DELX DELY DELZ

Status - OPTIONAL (if REFNAME was input)

Full Keyword - **TRANSLATE**

Function - Defines a translation to be applied to the reference surface in the creation of the current surface.

Input Variables -

DELX, DELY, DELZ (Real) - REQUIRED

Cartesian components of the translation.

Additional Information - None.

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE  SURF1
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFACE SURNEW SURF1
 TRAN   1.0  -1.0   0.0
 REFE   1.0   0.0   0.0
 DIRE   0.0   0.0   1.0
 ROTA   180.0
NORMAL 1001 +
```

REFE X Y Z

Status - OPTIONAL (if REFNAME was input)

Full Keyword - REFERENCE

Function - Defines a point on the axis defining the rotation.

Input Variables -

X,Y,Z (Real) - REQUIRED

Cartesian coordinates of a point on the axis of rotation.

Additional Information - None.

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE  SURF1
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFACE SURNEW SURF1
 TRAN   1.0  -1.0   0.0
 REFE   1.0   0.0   0.0
 DIRE   0.0   0.0   1.0
 ROTA   180.0
NORMAL 1001 +
```

DIRE   D1 D2 D3

Status - OPTIONAL (if REFNAME was input)

Full Keyword - DIRECTION

Function - Defines the positive direction of the axis of rotation.

Input Variables -

D1,D2,D3 (Real) - REQUIRED

Components of a vector along the positive direction of the axis of rotation.

Additional Information - None.

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE  SURF1
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFACE SURNEW SURF1
 TRAN   1.0  -1.0   0.0
 REFE   1.0   0.0   0.0
 DIRE   0.0   0.0   1.0
 ROTA   180.0
NORMAL 1001 +
```

ROTA THETA

Status - OPTIONAL (if REFNAME was input)

Full Keyword - ROTATION

Function - Defines the angle of rotation (in degrees) of the reference surface about the axis of rotation.

Input Variables -

THETA (Real) - REQUIRED

Angle of rotation (degrees).

Additional Information -

The angle of rotation is taken to be positive if it is counterclockwise when looking in the positive direction along the axis of rotation (right hand rule).

Examples of Use -

1. Define a 3-D surface called SURFNEW using the RENAME option which references a previously defined surface called SURF1.

```
SURFACE  SURF1
TYPE  QUAD
ELEMENT
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
SURFACE SURNEW SURF1
 TRAN   1.0  -1.0   0.0
 REFE   1.0   0.0   0.0
 DIRE   0.0   0.0   1.0
 ROTA   180.0
NORMAL 1001 +
```

`NORM    NEL1 F1 NEL2 F2 ... NELN FN`

Status - REQUIRED

Full Keyword - `NORMAL`

Function - Defines the outer normal direction on each disjoint boundary of the current GMR.

Input Variables -

NEL1 (Integer) - REQUIRED

User element number for an element in the current GMR.

F1 (Alphanumeric) - REQUIRED

Flag relating the outer normal direction on the element NEL1 to the input node ordering. Allowable values for F1 are + and − .

NEL2 ...NELN (Integer) - OPTIONAL

F2 ...FN -AL - OPTIONAL

Additional (element number, flag) pairs used to define the outer normal direction on disjoint boundaries of the current GMR, if any exist.

Additional Information -

A 'disjoint' boundary should not be confused with a 'surface'. A surface is merely a user convenience, permitting the collection of any number of elements without regard to connectivity. On the other hand, each disjoint boundary is the collection of contiguous elements. For example, the user could employ a single surface to model a thick-walled hollow sphere, but there will always be two disjoint boundaries.

The number of (element number, flag) pairs to be input for a given GMR is exactly equal to the number of disjoint boundaries which make up that GMR.

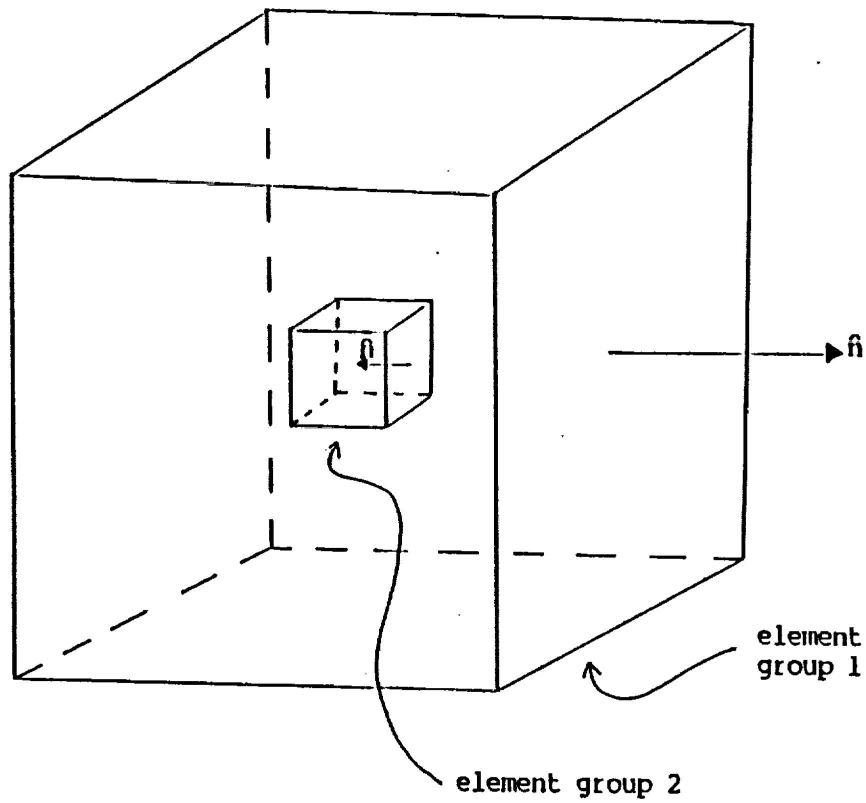
With the outer normal defined for one element on each disjoint boundary, the outer normals of all other elements in the GMR are automatically determined by the program.

If the input ordering of the nodes for element NEL1 (or NEL2 or NELN) is counterclockwise when viewed from outside the element, then F1 = +. Otherwise F1 = − (right hand rule).

Examples of Use -

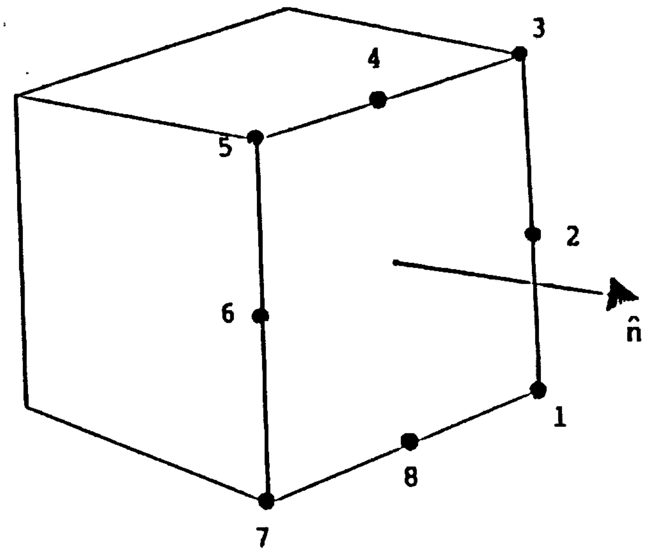
1. Define the direction of outward normal to the element 101 as being positive in relation to input node ordering.

```
**SURFACE  SIDE
  TYPE  QUAD
  ELEMENT
      101   1    2    3    103    203    202    201    101
      102   3    4    5    105    205    204    203    103
      103   5    6    7    107    207    206    205    105
      104   7    8    1    105    201    208    207    107
  NORMAL 101 +
```

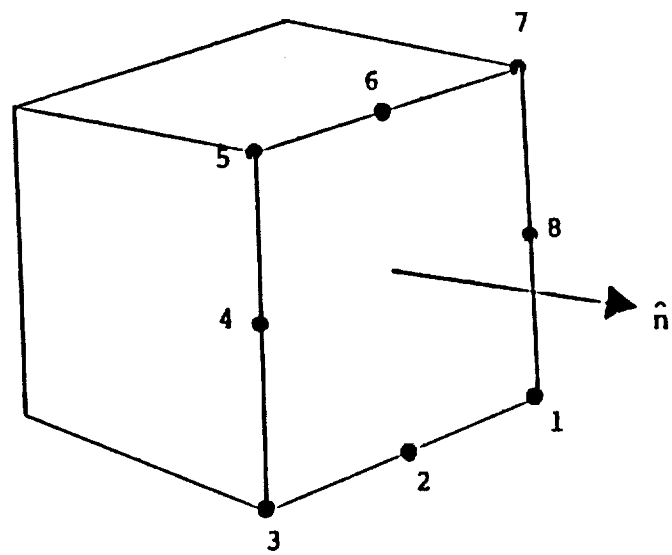


Element groups 1 and 2 are disjoint sections and therefore normals for one element in each section must be defined.

Figure for **GMR: NORM card
Three-dimensional disjoint (unconnected) elements



input shown is for
FLAG = +
on NORM card for 3-D



input shown is for
FLAG = -
on NORM card for 3-D

Figure for **GMR: NORM card

Three-dimensional outer normal convention
(In 3-D a mixed convention is allowed, as long as one
and only one normal per disjoint section is defined)

5.3.5 ENCLOSING ELEMENT DEFINITION

ENCL

Status - OPTIONAL

Full Keyword - `ENCLOSING`

Function - Signals the beginning of the connectivity definition for enclosing elements of the GMR. Enclosing elements are used in GMRs of infinite extent in order to create a fictitious boundary required for correct calculation of the matrix coefficients.

Input Variables - NONE

SIGN - (symbol) - REQUIRED

Additional Information -

In a GMR of infinite extent, it is neccessary to use either the EXTE or HALF card if enclosing elements are not used.

The nodes in an enclosing element do not become boundary source points (part of the system equation) unless they are also part of a regular boundary. The only purpose of enclosing elements is to define an arbitrary surface for integration so that the contribution of the unmodelled infinite boundary can be taken into account in the calculation of the diagonal terms of the F matrix (Rigid Body Translation Technique).

The geometry of the surface defined by the enclosing elements is arbitrary since the contribution (for a particular source point) of any surface enclosing the region is equivalent. Therefore, the discretization of enclosing elements should be crude, utilizing the minimum number of enclosing elements neccessary to enclose the region. It is, however, recommended that the surface defined by the enclosing elements does not pass too close (relative to the size of the enclosing element) to a boundary source point belonging to a regular element contained in that particular region.

In an analysis of a problem of a body of infinite extent, it is not neccessary to fix the boundary of the body for the sole purpose of preventing rigid body translation. Basically, the mathematics of the problem assumes zero displacement at infinity.

`(NONE)        NEL NODE1 ....   NODEN`

Status - REQUIRED (minimum of one card if ENCL is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the connectivity for a single enclosing element.

Input Variables -

NEL (Integer) - REQUIRED

User element number (required for user's purpose only)

NNODE (Integer) - REQUIRED

User number for the node for defining the geometry of the enclosing element.

NODEN = 8

Additional Information -

Only **EIGHT** noded elements are allowed.

All of the connectivity for enclosing elements must be defined such that their normals are positive.

Examples of Use -

1. Define enclosing elements

```
ENCLOSING ELEMENTS
 1001    1  2  3  103  203  202  201  101
 1002    3  4  5  105  205  204  203  103
 etc....
```

5.3.6 HOLE ELEMENT DEFINITION

HOLE

Status - OPTIONAL

Full Keyword - HOLE

Function - This card initiates the definition of embedded holes within the current GMR.

Input Variables - NONE

Additional Information -

In the present implementation, a hole is defined by a centerline and a radius. The centerline is represented by a two or three-noded line element. The three-noded element may be curved in space. The radius is constant along the length of the element.

Examples of Use -

1. Define a hole consisting of two quadratic hole elements.

```
HOLE
POINTS
  1001    0.5  0.5  0.0
  1002    0.5  0.5  0.25
  1003    0.5  0.5  0.5
  1004    0.5  0.5  0.75
  1005    0.5  0.5  1.0
TYPE  QUAD
ELEMENT
   101    0.1  1001   1002   1003
   102    0.1  1003   1004   1005
$(end of hole input)
```

POIN

Status - REQUIRED (if HOLE is input)

Full Keyword - POINTS

Function - This card initiates the definition of hole nodal points for the current GMR.

Input Variables - NONE

Additional Information -

Hole nodal points must be defined separately from the surface nodal points.

(NONE) NNODE X Y Z

Status - REQUIRED (if HOLE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - This card defines the node number and the Cartesian coordinates for hole node.

Input Variables -

NNODE (Integer) - REQUIRED
User number for the hole node.

X,Y,Z (Real) - REQUIRED Cartesian coordinates of the of the hole node.

Additional Information -

This card is input once for each point.

User node numbering must be unique within the problem. This includes surface, hole and interior points.

All point numbers must be less than or equal to 9999.

All hole nodes should be located on the interior of the part. Additionally, hole nodes should not be coincident with surface or sampling points.

Examples of Use -

1. Define a set of node numbers and the cartesian coordinates for nodes on two hole elements.

```
HOLE
POINTS
   101   1.0   0.0   1.0
   102   1.0   0.25  1.0
   103   1.0   0.5   1.0
   104   1.0   1.0   1.0
```

TYPE ETYPE

Status - REQUIRED (if HOLE is input)

Full Keyword - TYPE

Function - This card defines the variation of field quantities in the longitudinal direction for three-noded hole elements.

Input Variables -

ETYPE (Alphanumeric) - REQUIRED

Allowable values are LINE (linear shape function for field variables) or QUAD (quadratic shape function for field variables)

Additional Information -

The TYPE card defines the variation for three-noded hole elements only.

Only one TYPE card for holes is allowed per GMR.

All two-noded hole elements are assumed as linear longitudinal variation of the field quantities.

All of the three-noded hole elements in a single GMR must have the same type of variation.

This card has no effect on the circumferential variation of the field variables on the hole.

Examples of Use -

1. Specify that all three-noded hole elements appearing in the element list for hole elements will have a linear variation of field quantities in the longitudinal direction.

```
**HOLE
 POINTS
     101   1.0   0.0   1.0
     102   1.0   0.25  1.0
     103   1.0   0.5   1.0
     104   1.0   1.0   1.0
 TYPE LINEAR
   ELEMENT
     1001  0.2   101   102   103
     1002  0.2   103   104
```

ELEM

Status - REQUIRED (if HOLE is input)

Full Keyword - `ELEMENTS`

Function - Signals the beginning of the connectivity definition for hole elements in the current surface.

Input Variables - NONE

Additional Information - NONE

`(NONE)        NEL RADIUS NODE1 ...   NODEN`

Status - REQUIRED (if HOLE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the radius and connectivity for a single hole element.

Input Variables -

NEL (Integer) - REQUIRED

User hole element number.

RADIUS (Real) - REQUIRED

Radius of the hole.

NODE1 ... NODEN (Integer) - REQUIRED

User node numbers of the two or three hole nodes defining the geometry of the hole.

Additional Information -

This card is input once for each hole element.

The input card need not specify whether a two or three node element is being defined. This is determined strictly by the number of connectivity nodes included on the card.

User element numbers, including both surface and hole elements, must be unique within a problem.

All hole element numbers must be less than or equal to 9999.

All of the nodes referenced in the hole element connectivity must have been defined previously as POINTs under the HOLE keyword. That is, hole elements can only connect hole nodes. No surface nodes can be referenced by a hole element.

Two noded hole elements have a linear geometric representation, while three-noded elements use quadratic shape functions for their geometry.

The radius is assumed constant along the entire length of the hole element.

Examples of Use -

1. Define a set of data for two hole elements, specifying the element number, the radius and the connectivity. One hole element is three-noded with linear variation of field quantities and the other element is two-noded.

```
HOLE
POINTS
```

```
      101   1.0   0.0   1.0
      102   1.0   0.25  1.0
      103   1.0   0.5   1.0
      104   1.0   1.0   1.0
TYPE LINEAR
  ELEMENT
      1001  0.2   101   102   103
      1002  0.2   103   104
```

5.3.7 INSERT ELEMENT DEFINITION

INSE EVAL

Status - OPTIONAL

Full Keyword - `INSERT`

Function - This card initiates the definition of Insert inclusions within the current GMR and defines the Modulus of the Insert.

Input Variables -

EVAL (Alphanumeric) - REQUIRED
Elastic Modulus of the Insert.

Additional Information -

The Poisson's ratio of the insert is assumed to be the same as that of the GMR.

In the present implementation, an insert is defined by a centerline and a radius. The centerline is represented by a two or three-noded line element. The three-noded element may be curved in space. The radius is constant along the length of the element.

Examples of Use -

1. Define two inserts, one containing a single quadratic insert element, the other containing two quadratic insert elements.

```
 INSERT  35.0E+06
 POINTS
  1001  0.25  0.25  0.0
  1002  0.25  0.25  0.25
  1003  0.25  0.25  0.5
  1004  0.25  0.25  0.75
  1005  0.25  0.25  1.0
  2001  0.75  0.75  0.0
  2002  0.75  0.75  0.5
  2003  0.75  0.75  1.0
 TYPE  QUAD
  ELEMENT
  101   0.1  1001  1002  1003
  102   0.1  1003  1004  1005
  ELEMENT
  201   0.1  2001  2002  2003
$(end of insert input)
```

POIN

Status - REQUIRED (if INSE is input)

Full Keyword - `POINTS`

Function - This card initiates the definition of insert nodal points for the current GMR.

Input Variables - NONE

Additional Information -

Insert nodal points must be defined separately from the surface nodal points.

(NONE) NNODE X Y Z

Status - REQUIRED (if INSE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - This card defines the node number and the Cartesian coordinates for insert nodes.

Input Variables -

NNODE (Integer) - REQUIRED
User number for the insert nodes.

X,Y,Z (Real) - REQUIRED Cartesian coordinates of the of the insert nodes.

Additional Information -

This card is input once for each point.

User node numbering must be unique. This includes surface, insert and interior points.

All point numbers must be less than or equal to 9999.

All insert nodes should be located on the interior of the part. Additionally, insert nodes should not be coincident with surface or sampling points.

Examples of Use -

1. Define a set of node numbers and the cartesian coordinates for nodes on insert elements.

```
INSERT  30.0E+06
POINTS
    101   1.0   0.0   1.0
```

102	1.0	0.25	1.0
103	1.0	0.5	1.0
104	1.0	1.0	1.0
201	2.0	1.5	0.5
202	2.0	1.5	0.5
203	2.0	1.5	1.0

TYPE ETYPE

Status - REQUIRED (if INSE is input)

Full Keyword - TYPE

Function - This card defines the variation of field quantities in the longitudinal direction for three-noded insert elements.

Input Variables -

ETYPE (Alphanumeric) - REQUIRED

Allowable values are LINE (linear shape function for field variables) or QUAD (quadratic shape function for field variables)

Additional Information -

The TYPE card defines the variation for three-noded insert elements only.

Only one TYPE card for inserts is allowed per GMR.

All two-noded insert elements require linear longitudinal variation of the field quantities.

All of the three-noded insert elements in a single GMR must have the same type of variation.

This card has no effect on the circumferential variation of the field variables on the insert.

Examples of Use -

1. Specify that all three-noded insert elements appearing in the element list for insert elements have a quadratic variation of field quantities in the longitudinal direction.

```
INSERT  30.03E+06
POINTS
    101   1.0   0.0   10
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   1.0   1.0
    201   2.0   1.5   0.0
    202   2.0   1.5   0.5
    203   2.0   1.5   1.0
TYPE QUAD
```

ELEM

Status - REQUIRED (if INSE is input)

Full Keyword - `ELEMENTS`

Function - Signals the beginning of the connectivity definition for single insert with one or more insert elements.

Input Variables - NONE

Additional Information -

The ELEM card is repeated for each individual insert. An individual insert may contain more than one insert element which are connected end to end, however, an unconnected set of elements represents a set of inserts, in which each unconnected element must be preceded by the ELEM card.

(NONE) NEL RADIUS NODE1 ... NODEN

Status - REQUIRED (if INSE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Each card defines the radius and connectivity for a single insert element.

Input Variables -

NEL (Integer) - REQUIRED

User insert element number.

RADIUS (Real) - REQUIRED

Radius of the insert.

NODE1 ... NODEN (Integer) - REQUIRED

User node numbers of the two or three insert nodes defining the geometry of the insert.

Additional Information -

This card is input once for each insert element.

The input card need not specify whether a two or three node element is being defined. This is determined strictly by the number of connectivity nodes included on the card.

User element numbers, including both surface and insert elements, must be unique within the problem.

All insert element numbers must be less than or equal to 9999.

All of the nodes referenced in the insert element connectivity must have been defined previously as POINTs under the INSE keyword. That is, insert elements can only connect insert nodes. No surface nodes can be referenced by a insert element.

Two noded insert elements have a linear geometric representation, while three-noded elements use quadratic shape functions for their geometry.

The radius is assumed constant along the entire length of the insert element.

Examples of Use -

1. Define a set of data for three insert elements on two inserts specifying the element number, the radius and the connectivity. One insert contains one three-noded quadratic element and one two-noded linear element. The second insert contains one three-noded quadratic element.

```
INSERT  30.0E+06
POINTS
    101   1.0   0.0   1.0
    102   1.0   0.25  1.0
    103   1.0   0.5   1.0
    104   1.0   1.0   1,0
    201   2.0   1.5   0.0
    202   2.0   1.5   0.5
    203   2.0   1.5   1.0
TYPE QUAD
  ELEMENT
    1001  0.2   101   102   103
    1002  0.2   103   104
  ELEMENT
    2001  0.2   201   202   203
```

5.3.8 VOLUME CELL DEFINITION

VOLU NAME

Status - OPTIONAL

Full Keyword - VOLUME

Function - This card initiates the definition of a volume for the current GMR.

Input Variables -

NAME (Alphanumeric) - OPTIONAL
The name of the volume being defined. (For user's use only)

Additional Information -

In the present version of **BEST3D**, only one volume discretization per GMR is allowed. This means only one type (see next card definition) of cells, QUAD or LINE, can exist in a single GMR.

Hot spots and holes may be embedded within cells. Volume cells are not required if a global shape function discretization is used. It is only required for GMRs with thermal body force loading or when nonlinear effects are present.

Examples of Use -

1. Define three, 8-Noded linear volume cells.

```
VOLUME
TYPE  LINEAR
CELL
 1001  1  2  3  103  203  202  201  101
 1002  3  4  5  105  205  204  203  103
 1003  5  6  7  107  207  206  205  105
FULL
$(end of volume cell input)
```

TYPE TP

Status - REQUIRED (if VOLU is input)

Full Keyword - TYPE

Function - This card defines the variation of field quantities over the volume cells of the current GMR.

Input Variables -

TP (Alphanumeric) - REQUIRED

Allowable values are LINE and QUAD.

Additional Information -

A single GMR may contain either linear or quadratic cells, but not both.

If TP = QUAD is used and there are cells present, which are using linear geometry, then linear field variables will be assumed for those cells.

Examples of Use -

1. Specify that the variation of field quantities over the volume cells in GMR1 is quadratic in nature.

```
**GMR
  ID GMR1
  .
  .
  .
  VOLUME
  TYPE QUAD
```

CELL

Status - REQUIRED (if VOLU is input)

Full Keyword - CELLS

Function - Signals the beginning of the definition of volume cell input connectivity.

Input Variables - NONE

Additional Information -

Cell connectivity information is input on data cards following this card.

(NONE) NCELL N1 N2 .... NK

Status - REQUIRED (If VOLU is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Defines a volume cell in terms of previously defined nodal points.

Input Variables -

NCELL (Integer) - REQUIRED

User identification for cell being defined.

N1,N2,...,NK (Integer) - REQUIRED

User nodal point numbers for cell nodes.

NK = 4, 5, 6, 7, 8, 9, 10, 11, 15, 16, 20 or 21

Additional Information -

If necessary, this card may be input more than once for each cell. The cell number must be repeated on each card.

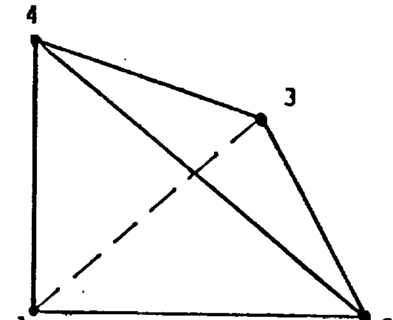
Cell numbering must begin at a corner and be numbered consecutively in either direction completing the face of a cell with the least number of nodes. The numbering of the remaining nodes begins at the same corner as the first face and continues in the same direction completing the mid-side nodes of the cell first (if present) and then the opposite face. If a central node is present, it is placed last.

Nodal points of the surface discretization may also be used in the volume discretization (ie. a cell face may match up with a boundary element). This is recommended when possible, since it somewhat reduces the computation required. Nodal points of the volume discretization lying on the surface need not, however, be nodal points of the surface mesh. In the event that such points do coincide, they can have different nodal point numbers in the surface and volume discretizations, although this should be avoided when possible.

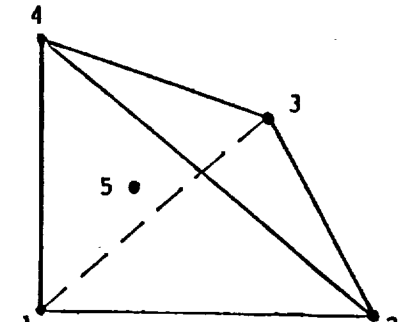
Examples of Use -

1. Define a set of volume cells consisting of a cell number and the connectivity information. There are three 8-noded volume cells with quadratic variation.

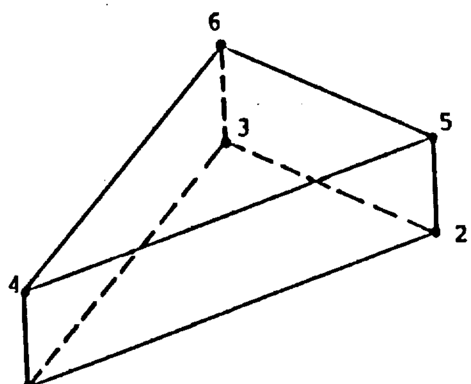
```
VOLUME
TYPE QUAD
CELL
    501   1   2   3   103   203   202   201   101
    502   3   4   5   105   205   204   203   103
    503   5   6   7   107   207   206   205   105
```



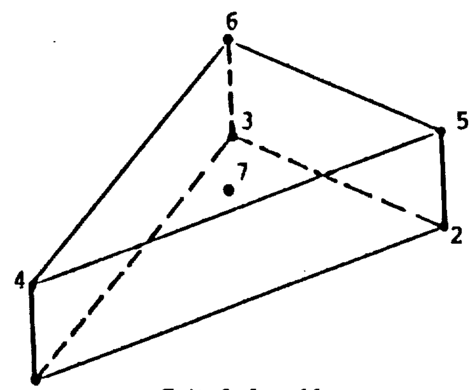
4-Noded Cell



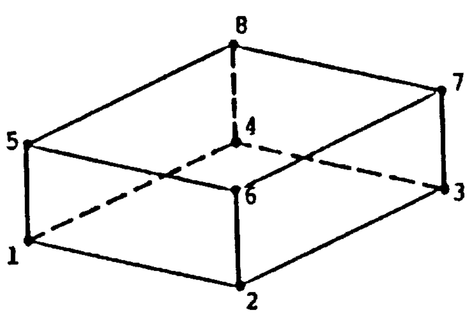
5-Noded Cell



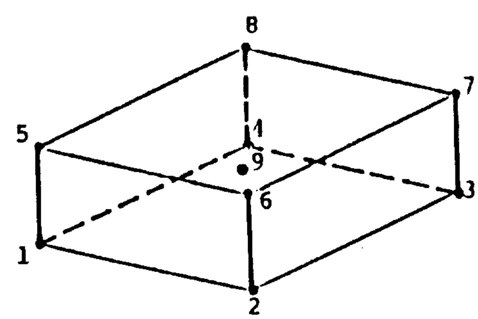
6-Noded Cell



7-Noded Cell



8-Noded Cell



9-Noded Cell

Figure for **GMR: Volume Cell Connectivity card
Linear, three-dimensional volume cells

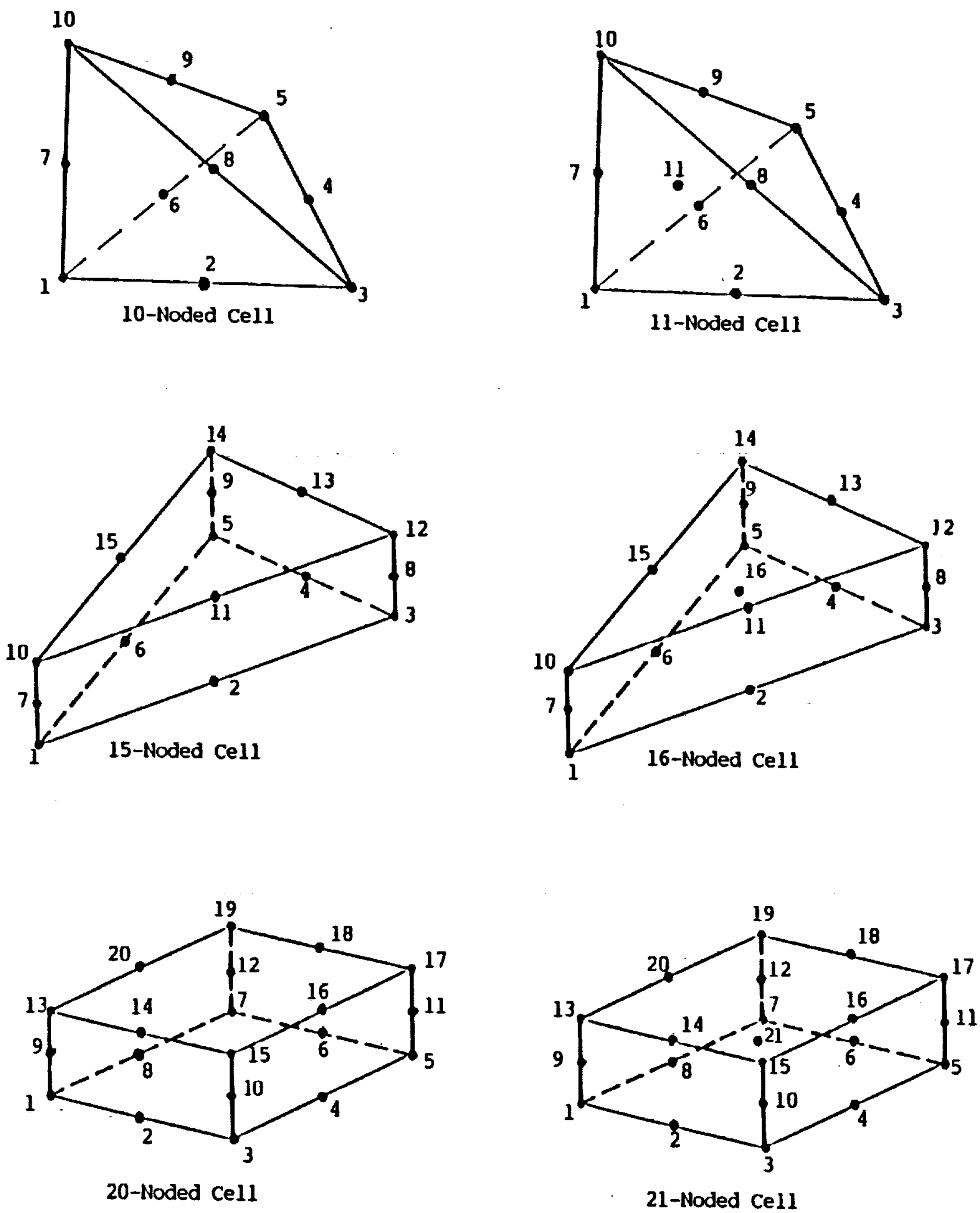


Figure for**GMR: Volume Cell Connectivity card
Quadratic, three-dimensional volume cells

FULL

Status - OPTIONAL

Full Keyword - `FULL` region of cells

Function - Identifies the GMR is completely filled with cells and that the Initial Stress Expansion Technique should be used to accurately calculate the coefficient corresponding to a singular point in the volume integration.

Input Variables - NONE

Additional Information -

The GMR must be completely filled by cells.

If FULL is not used, the relevant coefficients are calculated by volume integration complemented by the analytical jump term.

For highly accurate results, it is recommended that a GMR be completely filled with cells and that the FULL card option be exercised.

Examples of Use -

1. Specify that the GMR (GMR1) is completely filled with volume cells.

```
VOLUME
TYPE QUAD
CELL
  501   1   2   3   103   203   202   201   101
  502   3   4   5   105   205   204   203   103
  503   5   6   7   107   207   206   205   105
FULL
```

5.3.9 GLOBAL SHAPE FUNCTION DEFINITION

GLOB

Status - OPTIONAL

Full Keyword - `GLOBAL-SHAPE-FUNCTION-REPRESENTATION`

Function - Identifies a global shape function representation will be used in this GMR.

Input Variables - NONE

Additional Information -

> Not required if volume integration is used, and only required in GMR's with thermal load or which yielding occurs.

Examples of Use -

1. Define a 24-noded global shape function

```
 GLOBAL SHAPE FUNCTION
 NODE
 1  2  3  4  5  6  21  22  23  101  102  103
 104  105  201  202  203  301  302  303  401
 402  403  404
$ (end of global shape function input)
```

NODE

Status - OPTIONAL (If GLOB is input)

Full Keyword - `NODE LIST`

Function - Signals the beginning of a list of nodes that will be used for the global shape function.

Input Variables - NONE

Additional Information -

These nodes are the volume source points of the system. If a thermal gradient is present in the GMR, then a temperature data should be provided for these nodes in the Body Force Input Section. Otherwise these nodes are assumed to remain at the initial temperature of the GMR throughout the analysis.

A list of nodes to be used in the global shape function is input on data cards following the NODE card.

If NODE is not input all geometry points in the GMR will be used for the nodes of the global shape function. However, it is recommended that an explicit node list be input.

(NONE) N1 N2 ... NK

Status - REQUIRED (if NODE is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Defines the node list for the global shape function.

Input Variables -

N1,N2,...,NK (Integer) - REQUIRED

Nodes of the global shape function. Maximum of 20 nodes per line.

Additional Information -

Points for sampling points can be located anywhere in the GMR and these points are automatically added to the node list and therefore need not be specified on the node list.

This card can be repeated as many times as necessary with a maximum of 20 nodes per line and a maximum of 150 nodes per GMR (including interior points)

The nodal points may be contained in the regular nodal point definition for the GMR or on the hole or insert point list. These points may lie on the boundary or be contained in the interior of the region. Points in the interior

are recommended in order to produce an accurate representation of the interior distribution of the field variables.

No special ordering is necessary, however, ordering by proximity is preferred. Furthermore, a somewhat regular distribution of the global shape function nodes is recommended.

Sampling points need not be included in this list.

Examples of Use -

1. Specify a list of 24 nodes for a global shape function.

```
GLOBAL SHAPE FUNCTION
NODE
   1  2  3  4   5  6   21   22   23   101  102  103
   104  105  201  202  203  301  302  303  401  402
   403  404
```

5.3.10 SAMPLING POINT DEFINITION

SAMP

Status - OPTIONAL

Full Keyword - SAMPLING-POINTS

Function - This card signals the fact that a set of sampling points for which results are requested at any point on or in the body, will be provided for the current GMR.

Input Variables - NONE

ITYP1 (Alphanumeric) - REQUIRED

Additional Information -

This card is used to define points at which displacements, stresses, strains, temperatures, pressures and fluxes are to be calculated. The print flag for sampling points must be set in **CASE input. If, however, nothing is specified in **CASE for sampling points, this flag is set by default.

This card is followed by data cards defining the node number and coordinates of the sampling points.

Examples of Use -

1. Request result information at three interior points

```
SAMPLING-POINTS
 1001    0.333  0.25  0.0
 1002    0.25   0.1  10.0
 1003    0.2    0.5  20.2
```

(NONE) NNODE X Y Z

Status - REQUIRED (if SAMP is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Defines the coordinates of the sampling points for which output will be reported.

Input Variables -

NNODE (Integer) - REQUIRED
User number for the node.

X,Y,Z (Real) - REQUIRED Cartesian coordinates of the nodal point.

Additional Information -

This card is input once for each point.

User nodal point numbers must be unique, including the surface nodal points and any additional nodal points created for the volume discretization, or discretization of Holes and Inserts.

Point numbers must be less than or equal to 9999.

5.4 DEFINITION OF GMR COMPATIBILITY

When a body is modelled as an assembly of several GMRs suitable conditions must be specified to define the connections among the various regions. In the present version of **BEST3D** compatibility is defined between the interface surfaces of each pair of contacting regions. Four types of compatibility are allowed:

1 - Bonded contact : Continuity of all displacement components is imposed across the interface.

2 - Sliding contact : Continuity is required only for the component of displacement normal to the interface. The tractions, in both GMRs, in the tangent plane to the interface are set to zero.

3 - Spring or resistance contact : Spring or thermal resistance is imposed between regions.

4 - Cyclic contact : Symmetric elements within a cyclic symmetric part can have imposed symmetric deformation on these elements.

Continuity of temperature or pressure, when applicable, is imposed across the interface in either case.

A single nodal point location may be part of at most two GMRs. A single nodal point may be referenced in more than one interface definition data set as long as only two GMRs are involved. A single location must have a unique node number in each GMR. Various acceptable and unacceptable arrangements of GMRs are illustrated in figure for **INTE card.

The interface compatibility must be specified in such a way that there is one to one correspondence between the source points (field variable nodes) of the two GMR's that are involved. The input required to specify a single interface between two GMRs is described in the following pages, and a list of keywords recognized in the interface input are given below.

SECTION	KEYWORD	PURPOSE
5.4.1 Interface Definition Input Card		
	**INTE	Start of interface compatibility condition
5.4.2 Definition of interface surface 1		
	GMR	name of first GMR
	SURF	surface on first GMR
	ELEM	element of surface

SECTION	KEYWORD	PURPOSE
5.4.3 Definition of interface surface 2		
	GMR	name of second GMR
	SURF	surface on second GMR
	ELEM	element of surface
5.4.4 Type of interface conditions		
	BOND	bonded interface
	SLID	sliding interface
	SPRI	spring interface
	RESI	thermal resistence across interface
5.4.5 Cyclic Symmetry interface definition		
	CYCL	cyclic symmetry interface definition
	ANGL	angle for cyclic interface
	DIR	axis of rotation for cyclic interface
5.4.6 Additional Interface Control Options		
	TDIF	initial temperature difference across interface

5.4.1 INTERFACE DEFINITION INPUT CARD

**INTE

Status - OPTIONAL

Full Keyword - INTERFACE

Function - Indicates the beginning of an interface definition.

Input Variables - NONE

Additional Information -

A **INTE card must begin each interface definition. The complete definition of the connection between two GMRs may require more than one data set, since each data set can refer to only one surface.

The data set initiated with this card may be repeated as many times as required.

The interface surface reference below must be such that the nodes and elements of one GMR can be superimposed on the nodes and elements of the other GMR by translation and/or rotation, without any deformation.

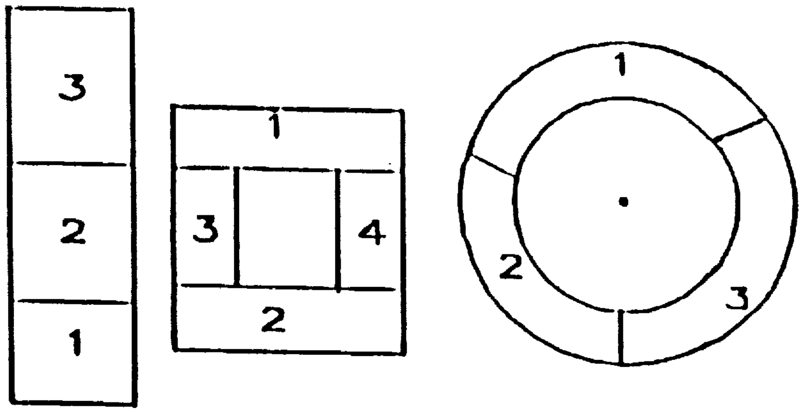
Note that each of the two GMR's involved in the interface definition must contain elements that lie on the interfacial surface.

The interface data sets must follow all GMR definitions, and must precede any boundary condition data sets.

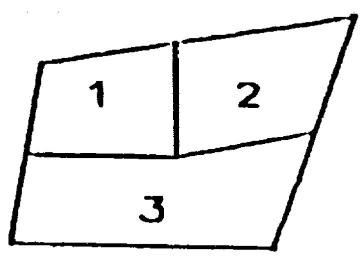
Examples of Use -

1. Defines the interface of two GMR's (default is perfectly bonded connection).

```
**INTERFACE
  GMR  REG1
     SURFACE  TOP
     ELEMENT  3  4  5
  GMR  REG2
     SURFACE  BOTTOM
     ELEMENT  103  104  105
```



Acceptable Connections



Unacceptable Connection

Figure for **INTE: card
Connections among GMRs

5.4.2 DEFINITION OF INTERFACE SURFACE 1

GMR IDGMR

Status - REQUIRED

Full Keyword - GMR

Function - Identifies the first GMR for which the interface surface is to be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometry definition (NAME on ID card in **GMR input).

Additional Information -

A given interface surface must lie entirely on the surface of a single GMR. If an interface compatibility condition is to be applied with more than one GMR, a separate interface compatibility must be defined for each case.

Examples of Use -

1. Identifies the first GMR, say GMR1, of which the interface surface is a part.

```
**INTERFACE
  GMR GMR1
    SURFACE  SURF1
    ELEMENTS 101    102    103    104
```

SURF IDSUR

Status - REQUIRED

Full Keyword - SURFACE

Function - Identifies the surface within the (first) selected GMR which embodies the interface surface (NAME on SURF card in **GMR input).

Input Variables -

IDSUR (Alphanumeric) - REQUIRED

Additional Information -

An interface surface must be contained entirely within a single surface. If the interface compatibility condition is to be applied to more than one surface, then a separate interface compatibility must be defined for each surface involved.

The SURF card may conclude the required input for a interface definition. If the SURF card is not followed by a ELEM card, then BEST3D will apply the interface compatibility condition to all of the elements in the surface IDSUR.

Examples of Use -

1. Identifies the interface surface, say SURF1, as part of the first GMR.

```
**INTERFACE
  GMR GMR1
    SURFACE  SURF1
    ELEMENTS 109    110
  GMR GMR2
    SURFACE  SURF2
```

ELEM EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - ELEMENTS

Function - Specifies the elements of the surface IDSUR to which an interface compatibility condition is to be applied.

Input Variables -

EL1,EL2,...,ELN (Integer) - REQUIRED

User element numbers of the elements of surface IDSUR which forms the interface surface.

Additional Information -

The effect of this card is to restrict the application of the compatibility condition to a portion of the surface IDSUR.

This input may be continued on more than one card. Each card must begin with the keyword ELEM.

If the ELEM card is specified, BEST3D will apply the interface compatibility condition only to the elements specified on this list.

In the present version of **BEST3D** , interface compatibility can not be specified at individual nodes.

Examples of Use -

1. Specifies three elements, 120, 121 and 122, for interfacial compatibility on the surface identified by the preceding SURFACE card.

```
**INTERFACE
  GMR  GMR1
    SURFACE  SURF1
    ELEMENTS 120   121   122
  GMR GMR2
```

5.4.3 DEFINITION OF INTERFACE SURFACE 2

GMR IDGMR

Status - REQUIRED

Full Keyword - GMR

Function - Identifies the second GMR for which the interface surface is to be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometry definition (NAME on ID card in **GMR input).

Additional Information -

A given interface surface must lie entirely on the surface of a single GMR. If an interface compatibility condition is to be applied with more than one GMR, a separate interface compatibility must be defined for each case.

Examples of Use -

1. Identifies the second GMR, say GMR2, of which the GMR surface is a part.

```
**INTERFACE
  GMR GMR1
    SURFACE  SURF1
    ELEMENTS 101    102
  GMR GMR2
    SURFACE  SURF2
    ELEMENTS 201    202
```

SURF IDSUR

Status - REQUIRED

Full Keyword - SURFACE

Function - Identifies the surface within the (second) selected GMR which embodies the interface surface (NAME on SURF card in **GMR input).

Input Variables -

IDSUR (Alphanumeric) - REQUIRED

Additional Information -

An interface surface must be contained entirely within a single surface. If the interface compatibility condition is to be applied to more than one surface, then a separate interface compatibility must be defined for each surface involved.

The SURF card may conclude the required input for a interface definition. If the SURF card is not followed by a ELEM card, then BEST3D will apply the interface compatibility condition to all of the elements in the surface IDSUR.

Examples of Use -

1. Identifies the interface, say SURF2, as part of the second GMR.

```
**INTERFACE
  GMR GMR1
    SURFACE   SURF1
    ELEMENTS 101    102
  GMR GMR2
    SURFACE   SURF2
    ELEMENTS 201    202
```

ELEM EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - ELEMENTS

Function - Specifies the elements of the surface IDSUR to which an interface compatibility condition is to be applied.

Input Variables -

EL1,EL2,...,ELN (Integer) - REQUIRED

User element numbers of the elements of surface IDSUR which forms the interface surface.

Additional Information -

The effect of this card is to restrict the application of the compatibility condition to a portion of the surface IDSUR.

This input may be continued on more than one card. Each card must begin with the keyword ELEM.

If the ELEM card is specified, BEST3D will apply the interface compatibility condition only to the elements specified on this list.

In the present version of **BEST3D** , interface compatibility can not be specified at individual nodes.

Examples of Use -

1. Specifies three elements, 210, 211 and 212, for interfacial compatibility on the surface identified by the preceding SURFACE card.

```
**INTERFACE
  GMR  GMR1
    SURFACE  SURF1
    ELEMENTS 101    102    103
  GMR  GMR2
    SURFACE SURF2
    ELEMENTS 210    211    212
**BCSET
```

5.4.4 TYPE OF INTERFACE CONDITION

BOND

Status - OPTIONAL

Full Keyword - BONDED

Function - Identifies a fully bonded interface.

Input Variables - NONE

Additional Information -

When this card is input continuity of all variables is imposed across the interface.

This is the default condition when the type of interface is not explicitly defined.

Examples of Use -

1. Defines a perfectly bonded interface of three boundary elements.

```
**INTERFACE
  BOND
  GMR  REG1
    ELEMENT   3   4   5
  GMR  REG2
    ELEMENT  103 104 105
```

SLID

Status - OPTIONAL

Full Keyword - SLIDING

Function - Identifies a sliding interface.

Input Variables - NONE

Additional Information -

When this card is input only normal displacement compatibility is imposed across the interface. The two GMRs are free to move in the plane tangent to the interface. This freedom may require the specification of additional boundary conditions to restrain rigid body motion.

Examples of Use -

1. Defines a sliding interface of five boundary elements.

```
**INTERFACE
 SLID
 GMR  REG1
   ELEMENT   101   102   103   104   105
 GMR REG2
   ELEMENT   210   212   213   214   215
```

SPRI IDIR KN

Status - OPTIONAL

Full Keyword - SPRING

Function - Identifies an interface with spring resistance in the normal direction between the corresponding surfaces. The normal tractions across this interface are linearly related to the difference in normal displacements between the two surfaces.

Input Variables -

IDIR (Integer) - REQUIRED

direction must be normal to the element (IDIR = 1)

KN (Real) - REQUIRED

Spring coefficient (K_n) normal to the element

Additional Information -

The SPRIng option utilizes the relationship $t_n = K_n(d_n^2 - d_n^1)$ where

d_n^1 normal displacement of GMR 1.

d_n^2 normal displacement of GMR 2.

t_n normal traction of GMR 1.

In the present version of the code a spring can only be defined in the direction normal to the element (i.e. IDIR = 1 only). In the direction tangential to the element a bonded relation is assumed unless the SLID card is also included.

The user is responsible for providing K_n in the proper units, consistent with the specifications of material properties, geometry, and boundary conditions.

The spring coefficient K_n should be a positive real number ($K_n > 0$). If zero is input, the coefficient will be automatically reset to 1.0E–10.

Examples of Use -

1. Defines a normal spring with a spring constant of 1.0E+03 units at the interface of two elements which are free to slide in the tangential direction.

```
**INTERFACE
  GMR  REG1
    SURFACE  TOP
    ELEMENT  101
  GMR REG4
    SURFACE  BOTTOM
    ELEMENT  401
  SPRING 1  1.0E+03
  SLIDING
```

RESI R1

Status - OPTIONAL

Full Keyword - RESISTANCE

Function - Identifies an interface with thermal resistance between the corresponding surfaces. The flux across this interface is linearly related to the temperature difference between the two surfaces.

Input Variables -

R1 (FP) - REQUIRED

Thermal resistance coefficient (R)

Additional Information -

The RESIstance option utilizes the relationship:

$$q_1 = \frac{1}{R1}(\theta_1 - \theta_2)$$

where

θ_1	local temperature of GMR 1.
θ_2	local temperature of GMR 2.
q_1	local heat flux from GMR 1.

The user is responsible for providing R in the proper units, consistent with the specification of material properties, geometry and boundary conditions.

The resistance R should be a positive real number ($R > 0$). If zero is input, the coefficient will be automatically reset to 1.0E–10.

Examples of Use -

1. Defines thermal resistance at the interface between two regions which were at the same initial temperature (otherwise, an extra card TDIF should be inserted between the RESI and **BCSET cards).

```
**INTERFACE
  GMR GMR1
    SURFACE  SURF1
    ELEMENTS 12
  GMR GMR2
    SURFACE  SURF2
    ELEMENTS 21
  RESI 1.0
**BCSET
```

5.4.5 CYCLIC SYMMETRY PARAMETER DEFINITION

CYCL

Status - OPTIONAL

Full Keyword - CYCLIC

Function - Identifies a cyclic symmetry boundary condition.

Input Variables - NONE

Additional Information -

This type of interface condition establishes a relationship between two boundary surfaces. In order for this condition to be applied the two boundary surfaces involved must be such that one can be exactly superimposed on the other by a rotation about a specified axis passing through the origin of the global coordinate system. Further, the imposed boundary conditions of the problem must be such that the deformed shape of one boundary surface can be exactly superimposed on the other by the same rotation. This option is intended for the analysis of (periodic) structures subjected to periodic loading.

CYCLE-A - Rigid body translation along the cyclic axis and rigid body rotation about that same axis are **not** automatically prevented by invoking the CYCLIC option. Consequently, these motions must be constrained explicitly by the user.

Since a cyclic interface condition involves all components of displacement and traction, **no other boundary condition may be applied to the elements that are involved.**

CYCLE-B - Local coordinate systems are established for each node on the second boundary surface. As a result, no other local system may be defined for these nodes. Furthermore, in the current version, it is recommended that displacement (or velocity) boundary conditions **not be applied** to any of the second surface nodes.

In the present version of **BEST3D**, a boundary surface to which a cyclic interface is applied may not intersect another interface.

A cyclic interface condition is time independent.

Examples of Use -

1. Activate option for cyclic symmetry boundary condition.

```
**INTERFACE
  GMR GMR1
  SURFACE SURF1
  ELEMENT 3
  GMR GMR1
  SURFACE SURF1
  ELEMENT 5
  CYCLIC
  ANGLE 20
  DIRECTION  0.  0.  1.
**BCSET
```

ANGL THETA

Status - REQUIRED (if CYCL is specified)

Full Keyword - **ANGLE**

Function - Specifies the angle of rotation between the two surfaces referenced in the cyclic symmetry condition.

Input Variables -

THETA (Real) - REQUIRED

THETA is the rotation angle (in degrees). A positive rotation is counterclockwise when looking along the positive axis direction.

Additional Information - NONE Examples of Use -

1. Specifies an angle of 20 degrees between the two surfaces referenced in the cyclic symmetry condition.

```
**INTERFACE
  GMR GMR1
  SURFACE SURF1
  ELEMENT 3
  GMR GMR1
  SURFACE SURF1
  ELEMENT 5
  CYCLIC
  ANGLE 20
  DIRECTION  0.  0.  1.
**BCSET
```

DIRE X Y Z

Status - OPTIONAL

Full Keyword - `DIRECTION`

Function - Defines the positive direction of the axis of rotation, if CYCL is specified.

Input Variables -

X,Y,Z (Real) - REQUIRED

Components of a vector along the positive direction of the axis of rotation.

Additional Information -

This card may be omitted. In this case the rotation axis defaults to the positive z-axis.

Examples of Use -

1. Defines that the positive direction of the axis of rotation is along the z-axis.

```
**INTERFACE
  GMR GMR1
  SURFACE SURF1
  ELEMENT 3
  GMR GMR1
  SURFACE SURF1
  ELEMENT 5
  CYCLIC
  ANGLE 20
  DIRECTION  0.  0.  1.
**BCSET
```

5.4.6 ADDITIONAL INTERFACE CONTROL OPTIONS

TDIF

Status - OPTIONAL

Full Keyword - TDIFFERENECE

Function - Signals that there is a difference in the initial temperatures of the two regions involved in the current interface.

Input Variables - NONE

Additional Information -

The TDIF card must be included in the interface definition for any temperature-dependent problem for which the initial temperatures of the adjoining regions are different. In such situations, failure to include this card will produce incorrect results.

It is expected that in future releases of **BEST3D**, the necessary checks will be done automatically, and the TDIF card will no longer be needed.

Examples of Use -

1. Indicates that a difference in initial temperatures exists between the GMR's, REG1 and REG2, involved in the current interface.

```
**INTERFACE
  GMR REG1
    SURFACE  TOP
    ELEMENTS 101    102
  GMR REG2
    SURFACE  BOTTOM
    ELEMENTS 209    210
  RESI  1.0
  TDIF
**BCSET
```

5.5 DEFINITION OF BOUNDARY CONDITIONS

This section describes the boundary condition input set (BCSET) for the input of boundary conditions applied at the surface of the given structure (or body). The input is designed to allow the specification of time dependent boundary conditions in both local and global coordinate systems. In order to allow the generality required, the input system is necessarily somewhat complex. Considerable simplification is possible for problems with less general requirements.

In the boundary element method, the primary load variable is traction (or flux), which acts over a surface area, not point forces (or sources) as in the finite element method. This means that in defining the region of application of a boundary condition in **BEST3D** it is necessary to specify both the nodal points and the elements involved.

A variety of options are provided for the definition of boundary conditions on the surface of the part. Each distinct set of boundary condition data defines either numerical values of variables over some portion of the surface of the part (or body), or establishes a relationship among variables. As many sets of boundary condition data may be used, as are required to completely specify the problem. A nodal point or element may be referenced in more than one set of boundary condition data.

A common process to much of the boundary condition input is the specification of the time dependent variables over the surface. To simplify the subsequent discussion of the various boundary condition types, the recurring definition of space/time variation is described only once in section 5.5.8.

<u>SECTION</u>	<u>KEYWORD</u>	<u>PURPOSE</u>
5.5.1 Boundary Condition Set Card		
	**BCSET	start of the B.C. definition
5.5.2 Boundary Condition Identification		
	ID	name of B.C. set
5.5.3 Identification of Boundary Condition Type		
	VALU	for specified B.C. value input
	RELA	for B.C. relation between boundary quantities
	LOCA	for local definition of B.C.

SECTION	KEYWORD	PURPOSE
5.5.4 Definition of Surface for Application of Boundary Conditions		
	GMR	identifies a GMR
	SURF	identifies the surface for this B.C. set
	ELEM	identifies surface elements
	POIN	identifies surface points
	TIME	defines the time for input
5.5.5 Value Boundary Condition for Surface Elements		
	TRAC	traction B.C. input
	DISP	displacement B.C. input
	FLUX	flux B.C. input
	TEMP	temp B.C. input
5.5.6 Definition of Hole for Application of Boundary Conditions		
	GMR	identifies a GMR
	HOLE	indicates B.C. on holes
	ELEM	identifies the hole elements
	POIN	identifies the hole points
	TIME	defines the time for input
5.5.7 Value Boundary Condition for Hole Elements		
	FLUX	flux B.C. input
	TEMP	temp B.C. input
	PRES	pressure B.C. input in hole
5.5.8 Definition of Space/Time variation		
	SPLI	source (field variable) point list
	T	nodal value of B.C.
5.5.9 Relation Boundary Condition		
	SPRI	spring relation (between displacement and traction)
	CONV	convection relation (between temperature and flux)

5.5.1 BOUNDARY CONDITION SET CARD

****BCSE**

Status - REQUIRED

Full Keyword - **BCSET

Function - Identifies the beginning of a boundary condition data set.

Input Variables - NONE

Additional Information -

As many boundary condition data sets may be input as are required. Each must begin with this card.

The boudary condition data sets must follow all GMR and INTERFACE definitions, and must precede any BODYFORCE data.

Examples of Use -

1. Fix the normal (local) displacement of all elements for on surface SIDE1 of gmr REG2 for all time (no TIME card required)

```
**BCSET
  ID U1FIX
  VALUE
  LOCAL
  GMR REG2
    SURFACE SIDE1
    DISP 1
    SPLIST ALL
    T 1 0.0
```

5.5.2 BOUNDARY CONDITION IDENTIFICATION

ID NAME

Status - REQUIRED

Full Keyword - ID

Function - Defines the identifier for the current boundary condition data set.

Input Variables -

NAME (Alphanumeric) - REQUIRED
User specified name of for the current data set.

Additional Information -

The NAME must be unique compared to all other boundary condition data set names defined in the problem.

The NAME must be eight or less alphanumeric characters. Blank characters embedded within the NAME are not permitted.

Examples of Use -

1. Define a set of displacement type boundary conditions with the name DISP1.

```
**BCSET
  ID  DISP1
  SURFACE  SURF1
  ELEMENTS  104
  POINT 108
  DISP 1
  SPLIST 108
  T   1   0.0
```

5.5.3 IDENTIFICATION OF BOUNDARY CONDITION TYPE

VALU

Status - OPTIONAL

Full Keyword - VALUE

Function - Identifies the boundary condition as one which will define the numerical values of field variables.

Input Variables - NONE

Additional Information -

This card must not be used for relational boundary condition sets.

If neither VALU nor RELA appears in a boundary condition set, a value-type set is assumed.

Examples of Use -

1. Used here to indicate that the value of a local traction type of boundry condition is specified.

```
**BCSET
  ID  TRAC12
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 17
    TRAC 1
  SPLIST ALL
    T 1   -100.0
$end of input data
```

RELA

Status - OPTIONAL

Full Keyword - RELATION

Function - Identifies the boundary condition as one which will define a relationship between field variables (e.g. spring or convection boundary conditions).

Input Variables - NONE

Additional Information -

This card is required for all boundary condition sets which define a relationship between field variables. Therefore, this card must be included for SPRING or CONVECTION boundary conditions.

Examples of Use -

1. The RELATION card is used in the following example to indicate specification of convection type of boundary condition.

```
**BCSET
  ID  BCS1
  RELATION
  GMR  GMR1
  SURFACE  SURF1
    ELEMENTS   1   2   3   4
    CONV  1.26   -100.0  $  H = 1.26, TEMP (AMBIENT) = - 100.0
**BCSET
  ID  BCS2
```

LOCA

Status - OPTIONAL

Full Keyword - `LOCAL`

Function - Indicates that input for the current boundary condition set will be in local coordinates.

Input Variables - NONE

Additional Information -

In the present version of **BEST3D** this option is intended for the specification of displacement, traction or spring constants normal to a (not necessarily plane) surface. Specification of conditions other than zero traction or flux in the tangent plane of the surface is unreliable.

In the local coordinate system the outer normal direction is the first coordinate direction.

Once a local boundary condition is specified on a node, the rest of the required boundary conditions on that node must be specified in local coordinates.

Once a local boundary condition is specified on a node of an element, the rest of the required boundary conditions on that element must be specified in local coordinates.

Local boundary conditions are not applicable to scalar problems (i.e. heat conduction).

5.5.4 DEFINITION OF SURFACE FOR APPLICATION OF BOUNDARY CONDITIONS

In the boundary element method, the primary load variable is traction (or flux), which acts over a surface area, not nodal forces (or sources) as in the finite element method. This means that in defining the region of application of a boundary condition in **BEST3D** it is necessary to specify both the nodal points and the elements involved.

If no boundary condition is specified (for a particular component) at a node, the primary load variable (of that component) is assumed to be zero.

The input lines involved in defining the element and nodes for a particular boundary condition set are described in this section.

GMR IDGMR

Status - REQUIRED

Full Keyword - GMR

Function - Identifies the GMR of the surface on which the boundary condition is to be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometry definition (NAME on ID card in **GMR input).

Additional Information -

A given boundary condition set can involve only a single GMR. If a boundary condition is to be applied to more than one GMR, a separate boundary condition set must be defined for each GMR.

Examples of Use -

1. Identifies the GMR name REG1 in connection with the specification of boundary conditions.

```
**BCSET
  ID  TRAC1
  VALUE
  GMR1  REG1
  SURFACE SURF1
  ELEMENTS   101   102
  TRAC  1
  SPLIST  ALL
  T  1  100.0
$ end of input data
```

SURF IDSUR

Status - OPTIONAL

Full Keyword - SURFACE

Function - Identifies the surface within the selected GMR on which the boundary condition is to be defined (NAME on SURF card in **GMR input).

Input Variables -

IDSUR (Alphanumeric) - REQUIRED

Additional Information -

Either this keyword or the HOLE keyword must be input for each boundary condition set.

A boundary condition set can involve only a single surface. If a boundary condition is to be applied to more than one surface, then a separate boundary condition set must be defined for each surface involved.

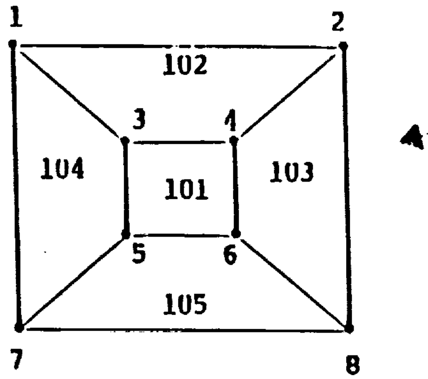
It is recommended that, whenever possible, surfaces be made to coincide with the regions over which boundary conditions are to be applied. This considerably simplifies the definition of surface for application of boundary condition.

If the SURF card is not followed by an ELEM or POIN card, then BEST3D will apply the boundary condition to all of the elements in the surface IDSUR.

Examples of Use -

1. Identifies the surface name SURF1 relevant to the specification of boundary conditions.

```
**BCSET
  ID  TRAC1
  GMR  GMR1
  SURFACE SURF1
  ELEMENTS  101   102   103
  TRAC  1
  SPLIST  ALL
  T  1  100.0
**BCSET
```



Input	Result
<u>Case 1 Input</u> GMR1 SURF1	Boundary condition applied at nodes 1 through 8. Over elements 101 through 105.
<u>Case 2 Input</u> GMR1 SURF1 ELEM 101	Boundary condition applied at nodes 3, 4, 5 and 6. Over element 101 only.
<u>Case 3 Input</u> GMR1 SURF1 ELEM 102 103 105 POIN 3 4 5 6	Boundary condition applied at nodes 3, 4, 5 and 6. Over elements 102, 103 and 105.
<u>Case 4 Input</u> GMR1 SURF1 ELEM 101 POIN 3	Boundary condition applied at node 3. Over element 101 only.

Figure for **BCSET: SURF, ELEM and POIN cards
3-D boundary subset definition

ELEM EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - ELEMENTS

Function - Specifies the elements of the surface IDSUR to which a boundary condition is to be applied.

Input Variables -

EL1,EL2,...,ELN (Integer) - REQUIRED

User element numbers of the elements of surface IDSUR which are to be included within the boundary condition set.

Additional Information -

The effect of this card is to restrict the application of the boundary condition to a portion of the surface IDSUR.

This input may be continued on more than one card. Each card must begin with the keyword ELEM.

If the ELEM card is not followed by a POIN card, then BEST3D will apply the boundary condition to all of the source points in the specified elements.

Examples of Use -

1. Specifies three elements on the surface SURF1 on which traction boundary conditions are given.

```
**BCSET
  ID  DISP2
  SURFACE  SURF1
  ELEMENTS  101    102    103
  DISP  2
  SPLIST  ALL
  T  1  0.0
```

`POIN P1 P2 ... PN`

Status - OPTIONAL

Full Keyword - `POINTS`

Function - Restricts the application of a boundary condition to a subset of the source points lying on the surface IDSUR.

Input Variables -

P1,P2,...,PN (Integer) - REQUIRED

Additional Information -

This card restricts the application of the boundary condition to the source points specified.

This card may be repeated as often as required. Each card must begin with the keyword.

If the POIN card is specified, then BEST3D will apply the boundary condition to all of the source points specified in this list. Also, if the POIN card is included, then the SPLIst card (see section 5.5.8) must explicitly list the same set of nodes.

Examples of Use -

1. Time-dependent input (in the x-direction) for points 5, 6, 7, and 8 over elements 102 and 103.

```
**BCSET
  ID BC1
  VALUE
  GMR REG3
    SURFACE BOTTOM
    ELEMENT 102 103
    POINT 5 6 7 8
    TIME 2.0 5.0 10.0
    TRAC 1
    SPLIST 6     7     5     8
    T 1 100.0 100.0 100.0 100.0
      T 2 200.0 200.0 300.0 300.0
      T 3 500.0 600.0 700.0 200.0
```

`TIME        T1 T2 ...   TN`

Status - OPTIONAL

Full Keyword - `TIMES`

Function - Specifies the times at which the variable involved in the boundary condition set will be specified.

Input Variables -

T1 (Real) - REQUIRED

First time point for boundary condition specification.

T2,...,TN (Real) - OPTIONAL

Subsequent time points for boundary condition specification.

Additional Information -

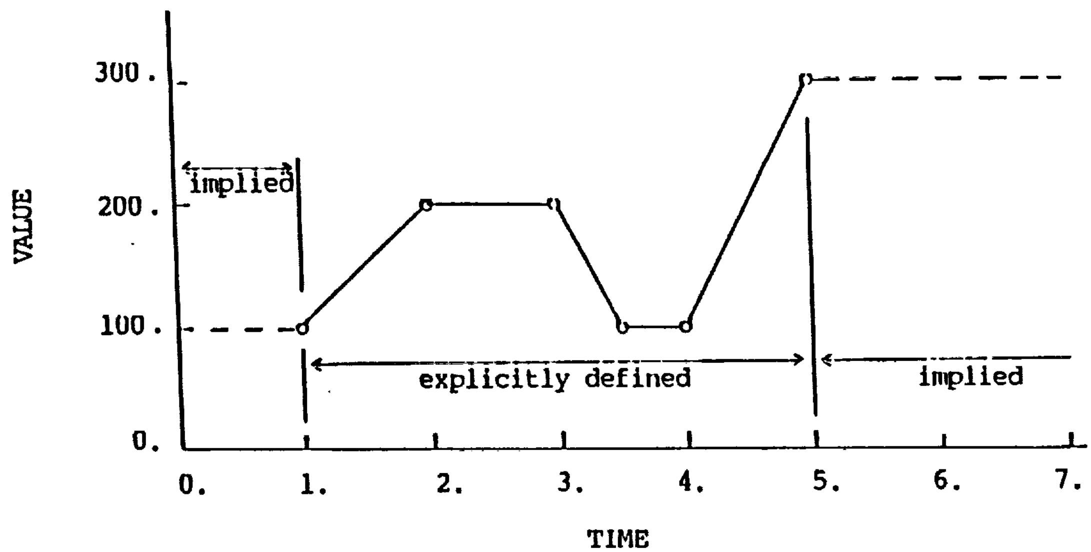
This input may be continued on more than one card if required. Each card must begin with the keyword TIME.

The time values input on this card need not agree with the times at which output was requested in the case control input. Different sets of time points may be used for different boundary conditions in the same analysis.

The time points must be specifed in ascending order.

Boundary condition values at other than input times are calculated by linear interpolation.

If a time card does not appear, the variables involved in the boundary condition are assumed to be time independent. Consequently, only a single time point may be specified for the SPACE/TIME VARIATION (as defined in section 5.5.8).



Example:

```
**BCSET
  ID TRAC1
  VALUE
    GMR GMR1
    SURF SURF1
    ELEMENT 3
      TIME 1.0 2.0 3.0 3.5 4.0 5.0
      TRAC 1
      SPLIST ALL
        T 1  100.0
        T 2  200.0
        T 3  200.0
        T 4  100.0
        T 5  100.0
        T 6  300.0
```

Figure for **BCSET: (all specified values)
VALUE vs. TIME for boundary condition input

5.5.5 VALUE BOUNDARY CONDITIONS FOR SURFACE ELEMENTS

TRAC IDIR

Status - OPTIONAL

Full Keyword - TRACTION

Function - Indicates that the IDIR component of traction will be specified for all nodes of the current boundary condition set.

Input Variables -

IDIR (Integer) - REQUIRED

Defines the component direction in which traction is specified. For cartesian coordinates:

1 - x direction

2 - y direction

3 - z direction

For local coordinates:

1 - outer normal direction

2,3 - must not be used (because local directions 2 and 3 are not unique)

Additional Information -

This card can only be used in a boundary condition data set containing the VALU card.

Up to three sets of traction and/or displacement specifications may be included in the same boundary condition data set. All must refer to the same boundary condition set. Only one condition (displacement or traction) may be applied in a given component direction.

The default condition is always to set traction to zero. After all boundary condition data sets have been processed, any boundary conditions not otherwise specified will be treated as zero traction conditions.

The TRACTION input line must be immediately followed by the space/time variation.

Examples of Use -

1. Defines a traction of magnitude 100.0 units in the positive y direction.

```
**BCSET
  ID TRAC1
  VALUE
  GMR  REG1
  SURFACE  SURF1
  ELEMENTS  105   106
  POINTS    112   113   114
  TRAC 2
  SPLIST    112   113   114
  T   1   100.0   100.0   100.0
```

Examples of Use -

2. Defines a traction of magnitude 100.0 units in the direction of the outward normal.

```
**BCSET
  VALUE
  LOCAL
  GMR  GMR1
  SURFACE SURF1
  ELEMENTS  101  102
  TRAC 1
  SPLIST ALL
  T  1  100.0
$ end of data set
```

DISP IDIR

Status - OPTIONAL

Full Keyword - DISPLACEMENT

Function - Indicates that the IDIR component of displacement will be specified for all nodal points contained in the current boundary condition set.

Input Variables -

IDIR (Integer) - REQUIRED

Defines the component direction in which displacement is specified. For global coordinates:

1 - x direction

2 - y direction

3 - z direction

For local coordinates:

1 - outer normal direction

2,3 - must not be used (because local directions 2 and 3 are not unique)

Additional Information -

This card can only be used in a boundary condition data set containing the VALU card.

Up to three sets of traction and/or displacement specifications may be included in the same boundary condition data set. All must refer to the same boundary condition set. Only one condition (displacement or traction) may be applied in a given component direction.

The default condition is always to set traction to zero. After all boundary condition data sets have been processed, any boundary conditions not otherwise specified will be treated as zero traction conditions.

The DISPLACEMENT input line must be immediately followed by space/time variation.

Examples of Use -

1. Defines complete fixity of nodes, (i.e., displacements in x, y and z directions are zero) of three elements.

```
**BCSET
```

```
  ID  DISP1
  VALUE
  GMR  GMR1
  SURFACE  SURF1
  ELEMENTS  104  105  106
  DISP  1
  SPLIST  ALL
  T  1  0.0
  DISP 2
  SPLIST  ALL
  T  2  0.0
  DISP 3
  SPLIST  ALL
  T  3  0.0
**BCSET
  ID  TRAC1
```

FLUX

Status - OPTIONAL

Full Keyword - `FLUX`

Function - Indicates that the flux will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

FLUX is only valid for HEAT analysis.

This card can only be used in a boundary condition data set containing the VALU card.

The specification of flux may be included with up to three sets of traction and/or displacement specifications in the same boundary condition data set for concurrent analyses. All must refer to the same boundary condition set.

When applicable, the default condition is to set flux to zero.

The FLUX input line must be immediately followed by the space/time variation.

Examples of Use -

1. Defines zero flux conditions across three elements.

```
**BCSET
  ID  ENTER1
  VALUE
  GMR  GMR1
  SURFACE  SURF1
  ELEMENTS  22  23  24
  FLUX
  SPLIST  ALL
  T  1  0.0
```

TEMP

Status - OPTIONAL

Full Keyword - TEMPERATURE

Function - Indicates that the temperature will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

TEMP is only valid for HEAT analysis.

This card can only be used in a boundary condition data set containing the VALU card.

The specification of temperature may be included with up to three sets of traction and/or displacement specifications in the same boundary condition data set for concurrent analyses. All must refer to the same boundary condition set.

When applicable, the default condition is to set flux to zero.

The TEMP input line must be immediately followed by the space/time variation.

Examples of Use -

1. Indicates that a constant temperature is specified on the relevant elements.

```
**BCSET
  ID  TOP
  VALUE
  GMR  GMR2
  SURFACE  SURF2
  ELEMENTS  218  219
  VELO  1
  SPLIST  ALL
  T  1  0.0
  TEMP
  SPLIST ALL
  T  1  0.0
```

5.5.6 DEFINITION OF HOLES FOR APPLICATION OF BOUNDARY CONDITIONS

GMR IDGMR

Status - REQUIRED

Full Keyword - GMR

Function - Identifies the GMR on the surface of which the boundary condition is to be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometry definition (NAME on ID card in **GMR input).

Additional Information -

A given boundary condition set can involve only a single GMR. If a boundary condition is to be applied to more than one GMR, a separate boundary condition set must be defined for each GMR.

Examples of Use -

1. Define a linear increasing pressure inside a hole (elements 203, 204) for stress analysis.

```
**BCSET
  GMR  REG1
  HOLE
  ELEMENT  203  204
  TIME  0.0  10.0
    PRESSURE
    SPLIST  ALL
    T  1  0.0
    T  2  100.0
```

HOLE

Status - OPTIONAL

Full Keyword - HOLE

Function - Identifies this as a boundary condition set for hole elements within the selected GMR.

Input Variables - NONE

Additional Information -

Either this keyword or the SURF keyword must be input for each boundary condition set.

If the HOLE keyword line is not followed by an ELEM line, then **BEST3D** will apply the boundary condition to all of the hole elements in the current GMR.

Examples of Use -

1. Heat transfer – Define a temperature (for all time) at points in elements 509, 510.

```
**BCSET
  GMR  GMR5
    HOLE
    ELEMENT  509
    POINTS  5025  5026  5027
      PRESSURE
      SPLIST  5025   5025   5027
      T  1    100.0  150.0  200.0
```

ELEM EL1 EL2 ... ELN

Status - OPTIONAL

Full Keyword - ELEMENTS

Function - Specifies the hole elements to which a boundary condition is to be applied.

Input Variables -

EL1,EL2,...,ELN (Integer) - REQUIRED

User element numbers of the hole elements which are to be included within the boundary condition set.

Additional Information -

The effect of this card is to restrict the application of the boundary condition to a group of hole elements within the previously specified GMR.

This input may be continued on more than one card. Each card must begin with the keyword ELEM.

If the ELEM card is not followed by a POIN card, the **BEST3D** will apply the boundary condition to all of the source points in the specified hole element.

Examples of Use -

1. Define a flux of 100 for points in elements 207, 208, 209 and 311 (for all time).

```
**BCSET
  GMR REG5
    HOLE
    ELEMENT  207  208  209  311
      FLUX
      SPLIST ALL
      T 1 100.0
```

```
POIN      P1 P2 ...  PN
```

Status - OPTIONAL

Full Keyword - POINTS

Function - Restricts the application of a boundary condition to a subset of the source points associated with hole elements specified above.

Input Variables -

P1,P2,...,PN (Integer) - REQUIRED

Additional Information -

This card restricts the application of the boundary condition to the nodal hole points specified.

This card may be repeated as often as required. Each card must begin with the keyword.

If the POIN card is specified, then BEST3D will apply the boundary condition to all of the hole points specified in this list. Also, if the POIN card is included, then the SPLIst card (see section 5.5.8) must explicitly list the same set of hole nodes.

Examples of Use -

1. Define a linear increasing pressure (in time) that varies from point to point in an element.

```
**BCSET
  GMR REG9
    HOLE
    ELEMENT 207
    POINTS  2005  2007  2011
    TIME    0.0   1.0
      PRESSURE
      SPLIST  2005  2007  2011
        T  1   0.0   0.0   0.0
        T  2   10.0  15.0  20.0
```

```
TIME        T1 T2 ...   TN
```

Status - OPTIONAL

Full Keyword - TIMES

Function - Specifies the times at which the variable involved in the boundary condition set will be specified.

Input Variables -

T1 (Real) - REQUIRED

First time point for boundary condition specification.

T2,...,TN (Real) - OPTIONAL

Subsequent time points for boundary condition specification.

Additional Information -

This input may be continued on more than one card if required. Each card must begin with the keyword TIME.

The time values input on this card need not agree with the times at which output was requested in the case control input. Different sets of time points may be used for different boundary conditions in the same analysis.

The time points must be specified in ascending order.

Boundary condition values at other than input times are calculated by linear interpolation.

If a time card does not appear, the variables involved in the boundary condition are assumed to be time independent. Consequently, only a single time point may be specified for the SPACE/TIME VARIATION (as defined in section 5.5.8).

Examples of Use -

1. Define a time-dependent (4 time points) temperature for two points in element 907.

```
**BCSET
  GMR REG10
    HOLE
    ELEMENT  907
    POINTS   9003  9004
    TIME     0.0   5.0   10.0   20.0
      TEMPERATURE
      SPLIST  9003  9004
      T   1   0.0   0.0
      T   2   10.0  20.0
      T   3   20.0  50.0
      T   4   30.0  60.0
```

5.5.7 VALUE BOUNDARY CONDITIONS FOR HOLE ELEMENTS

PRES

Status - OPTIONAL

Full Keyword - PRESSURE

Function - Indicates that the pressure will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

This card can only be used in a boundary condition data set containing the VALU card.

When applicable, the default condition is to set pressure to zero.

The PRESSURE input line must be immediately followed by the space/time variation.

Examples of Use -

1. Stress Analysis - Define a time dependent pressure inside a hole (element 101).

```
**BCSET
  GMR  GMR5
    HOLE
    ELEMENT 101
    TIME  0.0  5.0  10.0  2.0
      PRESSURE
      SPLIST ALL
      T 1 0.0
      T 2 50.0
      T 3 500.0
      T 4 1000.0
```

FLUX

Status - OPTIONAL

Full Keyword - FLUX

Function - Indicates that the flux will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

FLUX is only valid for HEAT analysis.

This card can only be used in a boundary condition data set containing the VALU card.

When applicable, the default condition is to set flux to zero.

The FLUX input line must be immediately followed by the space/time variation.

Examples of Use -

1. Define a flux (for all times) at points in element 509.

```
**BCSET
  GMR REG3
    HOLE
    ELEMENT  509
    POINT    5003  5004
      FLUX
      SPLIST 5003  5004
      T  1   5.0   6.0
```

TEMP

Status - OPTIONAL

Full Keyword - TEMPERATURE

Function - Indicates that the temperature will be specified for all nodes of the current boundary condition set.

Input Variables - NONE

Additional Information -

TEMP is only valid for HEAT analysis.

This card can only be used in a boundary condition data set containing the VALU card.

When applicable, the default condition is to set flux to zero.

The TEMP input line must be immediately followed by the space/time variation.

Examples of Use -

1. Heat Transfer - Define a time-dependent temperature inside a hole.

```
**BCSET
  GMR EXCHANGER
    HOLE
    ELEMENT 201
    TIME    0.0  5.0  10.0  3.0
      TEMPERATURE
      SPLIST
      T 1 0.0
      T 2 20.0
      T 3 50.0
      T 4 100.0
```

5.5.8 DEFINITION SPACE/TIME VARIATION

```
SPLI      N1 N2 ...  NN
```

Status - REQUIRED

Full Keyword - `SPLIST (source point list)`

Function - Defines the order in which nodal values of the variable will be input.

Input Variables -

N1 (Integer or Alphanumeric) - REQUIRED
User nodal point number of first node for which data will be input. Optional values are ALL or SAME, described under Additional Information.

N2,...,NN (Integer) - REQUIRED (if ALL or SAME are not used)
Users nodal point number of all remaining nodes that are defined by the definition of surface for application of Boundary Conditions (section 5.5.4).

Additional Information -

This input may be continued on more than one card if required. Each card must begin with the keyword SPLI.

If N1 = ALL, then **BEST3D** assigns the same value of the input variable to all nodes defined by the definition of surface for application of Boundary Condition (section 5.5.4).

If N1 = SAME, then the nodal point ordering is taken to be the same as that defined for the immediately preceding boundary condition specification within the same boundary condition set. N1 = SAME may not be used for the first boundary condition specification within a boundary condition set.

If the node number input is used (i.e. if ALL or SAME are not used) then the **total number of points in SPLI must equal the number of nodes defined by the definition of surface for application of Boundary Conditions (section 5.5.4).**

T IT V1 V2 ... VN

Status - REQUIRED

Full Keyword - T

Function - Identifies a data card containing values of a variable specified in a boundary condition at time point IT.

Input Variables -

IT (Integer) - REQUIRED

Time point as specified on the TIME card in the definition of the surface for application of boundary condition (section 5.5.4). IT = 1 refers to the first time point, IT = 2 the second, etc.

V1,V2,...,VN (Real) - REQUIRED

Nodal values of the variable in the nodal point order defined on the SPLI card.

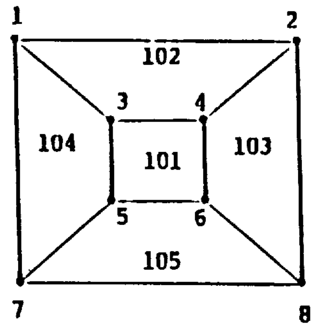
Additional Information -

This input may continue for as many cards as required. Each card defining the values at time point IT must begin with T. The input for each new time point must begin on a new card.

If N1 = ALL or SAME on the SPLI card, then only a single value of the variable is input for each time point.

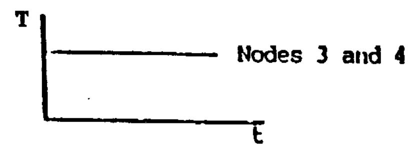
If the card "LOAD COMP" is specified in the case input, i.e. the boundary conditions are complex values, then the real part and imaginary part of nodal values are input in the form V1R V1I V2R V2I ...... VNR VNI. More cards can be added, if one card is not enough to specify the needed boundary conditions.

The results of various uses of the SPLI and T cards are shown in the figure on the following page.

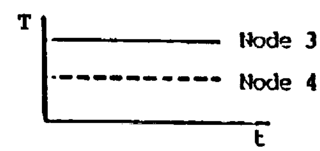


Note differences in traction input of nodes 3 and 4 over elements 101 and 102

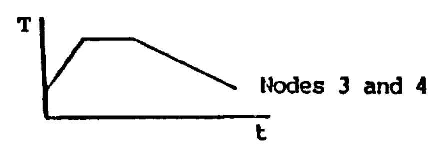
```
Case 1 Input
TIME   0.0
TRAC 1
SPLIST ALL
T 1  100.0
```




```
Case 2 Input
TIME   0.0
TRAC 1
SPLIST 3  4
T 1  100.0  50.0
```




```
Case 3 Input
TIME   0.0  1.0  2.0  4.0
TRAC 1
SPLIST ALL
T 1  50.0
T 2  150.0
T 3  150.0
T 4  50.0
```




```
Case 4 Input
TIME   0.0  1.0  2.0  4.0
TRAC   1
SPLIST 3  4
T 1  75.0   50.0
T 2  100.0  25.0
T 3  75.0   50.0
T 4  75.0   25.0
```

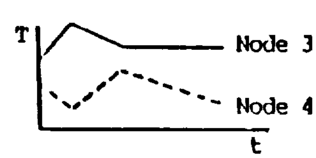


Figure for BCSET: TIME and SPLIST cards

TIME-SPACE variation

5.5.9 RELATION BOUNDARY CONDITION

SPRI IDIR CONST

Status - OPTIONAL

Full Keyword - SPRING

Function - Identifies a boundary condition in which a specified component of displacement and traction are linearly related by a spring constant for all nodal points defined in the current boundary condition.

Input Variables -

IDIR (Integer) - REQUIRED

Defines the component direction in which the relationship between displacement and traction is specified. For global coordinates:

1 - x direction

2 - y direction

3 - z direction

For local coordinates:

1 - outer normal direction

2,3 - must not be used (because local directions 2 and 3 are not unique)

CONST (Real) - REQUIRED

Spring constant (for steady-state forced vibration problems, it should consist of both real and imaginary parts of the spring constant).

Additional Information -

The SPRI card can only be used if the RELA card has been input for the current boundary condition data set.

This option is intended primarily to allow a boundary condition that accounts for the response of other structures which are not modelled in **BEST3D**, through their stiffnesses (i.e. spring constants).

Currently, only time independent spring boundary conditions are available.

The real portion of the spring constant should be positive. If zero is input, the real portion will be automatically reset to 1.0E-10.

Examples of Use -

1. Specifies a spring of constant 1.0E+03 units in the x direction.

```
**BCSET
  ID  BCS1
  RELATION
  GMR GMR1
  SURFACE SURF1
    ELEMENTS  1   2  3  4
    SPRING    1   1.0E+03
**BCSET
  ID BCS2
```

CONV FCOEFF TAMBT

Status - OPTIONAL

Full Keyword - CONVECTION

Function - Identifies a boundary condition in which surface temperature minus ambient temperature is linearly related to flux via a film coefficient for all nodal points defined in the current boundary condition.

Input Variables -

FCOEFF (Real) - REQUIRED

Convective film coefficient (h)

TAMBT (Real) - OPTIONAL

Ambient temperature of convective fluid (T_a)

Additional Information -

CONV is only valid for HEAT analysis.

The CONV card can only be used if the RELA card has been input for the current boundary condition data set.

The CONVection option utilizes the relationship:

$$Q = -H * (T_a - T)$$

The film coefficient must be time independent.

If a TIME card was **not** included in the current boundary condition set, then the ambient temperature is time independent and TAMBT must be specified in the CONV card.

If a TIME card was included in the current BCSET, then T card(s) must follow the CONV card to define the time variation of ambient temperature.

No spatial variation of film coefficient nor ambient temperature is permitted within an individual BCSET.

The film coefficient should be set to a positive value. If zero is input, the coefficient will be automatically reset to 1.0E–10.

Examples of Use -

1. Defines a film coefficient of 1.26 units and a surface to ambient temperature difference of 100 units.

```
**BCSET
  ID  BCS1
```

```
  RELATION
  GMR  GMR1
  SURFACE SURF1
    ELEMENTS  1  2  3  4
    CONV      1.26  -100.0
**BCSET
  ID  BCS2
```

5.6 BODY FORCE DEFINITION

This section describes the input for body forces.

The following body forces are included in **BEST3D**: thermal, hot spot, centrifugal and inertial loads. The input cards required to define these loads are described below.

SECTION	KEYWORD	PURPOSE
5.6.1 Body Force Input Card		
	**BODY	start of body force input
5.6.2 Centrifugal body force		
	CENT	centrifugal load input
	DIRE	direction of axis of rotation
	POINT	point on the axis of rotation
	TIME	time for input
	SPEE	speed input
5.6.3 Inertial body force		
	INER	inertial body force input
	DIRE	direction of acceleration
	TIME	time of input
	ACCE	accleration input
5.6.4 Thermal Body Force		
	THER	thermal body force input
	TIME	times for input
	GMR	Identifies GMR
	TEMP	start of temperature input for GMR
	(node temperatures)	
5.6.5 Embedded Hot Spot		
	HOT	hot spot input
	TIME	time for input
	GMR	Identifies GMR
	TEMP	start of hot spot temperature input for GMR
	(node temperatures)	

5.6.1 BODY FORCE INPUT CARD

**BODY

Status - OPTIONAL

Full Keyword - BODY FORCE

Function - Identifies the beginning of body force input.

Input Variables - NONE

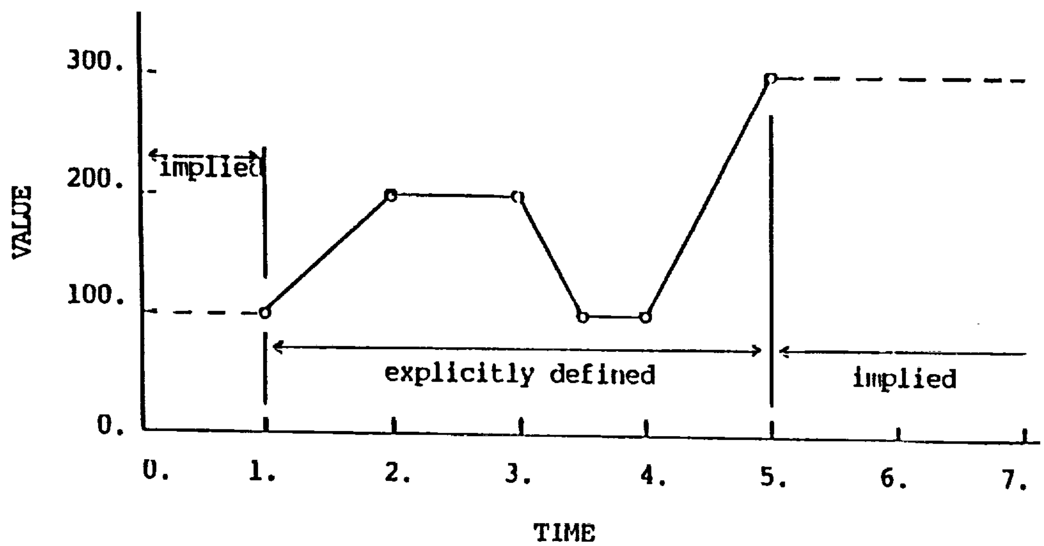
Additional Information -

> If more than type of body force is present, a separate block starting with **BODY should be defined for each type.

Examples of Use -

1. Request a three-dimensional centrifugal and thermal input.

```
**BODY FORCE
  CENT
  DIRE  0.0  0.0  1.0
  POINT 0.0  0.0  0.0
  TIME  1.  2.  3.  4.
  SPEED  45.  80.  100.  120.

**BODY FORCE
  THER
  TIME  0.  5.
  GMR  REG1
  TEMP
    1   0.0  500.
    2   0.0  500.
    3   0.0  300.
```



Example:

```
**BODY FORCE
  CENTRIFUGAL
    POINT   0.0   0.0
    TIME    1.0   2.0   3.0   3.5   4.0   5.0
    SPEED 100.0 200.0 200.0 100.0 100.0 300.0
```

Figure for**BODY: (all body force values)
VALUE vs. TIME for body force input

5.6.2	CENTRIFUGAL BODY FORCE

CENT

Status - OPTIONAL

Full Keyword - `CENTRIFUGAL`

Function - Indicates that a centrifugal load will be applied.

Input Variables - NONE

Additional Information -

Only one (time dependent) centrifugal load condition may be defined for an analysis. It is applied to the entire part.

Examples of Use -

1. Request a three-dimensional centrifugal input.

```
**BODY FORCE
 CENT
 DIRE  0.0  0.0  1.0
 POINT  0.0  0.0  0.0
 TIME   1.    2.    3.
 SPEED  50.   150.   200.
```

DIRE X Y Z

Status - OPTIONAL

Full Keyword - DIRECTION

Function - Defines a vector parallel to the axis of rotation.

Input Variables -

X,Y,Z (Real) - REQUIRED

Cartesian components of a vector parallel to the axis of rotation.

Additional Information -

Only one direction can be defined in an analysis

If this card is omitted, the axis of rotation is assumed to be parallel to the z-axis of the global system.

Examples of Use -

1. Request a three-dimensional centrifugal input with the direction of rotation parallel to the X-axis.

```
**BODY FORCE
    CENT
    DIRE   1.0  0.0  0.0
    POINT  0.0  0.0  0.0
    TIME   1.0
    SPEED  100.
```

POIN XP YP ZP

Status - OPTIONAL

Full Keyword - POINT

Function - Defines a reference point on the axis of rotation.

Input Variables -

XP,YP,ZP (Real) - REQUIRED

Cartesian coordinates of a point on the axis of rotation.

Additional Information -

If this card is omitted, the reference point is taken to be the origin (0,0,0) in the global system.

Examples of Use -

1. Request a three-dimensional centrifugal input for rotation about an axis centered between the points (1,1,0) and (5,5,1)

```
**BODY FORCE
  CENT
  DIRE   4.  4.  1.
  POINT  1.  1.  0.
  TIME   1.0
  SPEED  50.0
```

```
TIME     T1 T2 ...   TN
```

Status - REQUIRED (if CENT is input)

Full Keyword - `TIMES`

Function - Defines the times at which the speed of rotation of the part will be defined.

Input Variables -

T1,T2,...,TN (Real) - REQUIRED

Times at which speed of rotation will be defined.

Additional Information -

This card may be input as many times as required. Each card begins with the keyword TIME.

A maximum of 20 time values may be specified.

Examples of Use -

1. Request an input for three-dimensional centrifugal input at 5 times.

```
**BODY FORCE
  CENT
  DIRE   0.0  0.0  1.0
  POINT  0.0  0.0  0.0
  TIME   0.0  2.0  4.0  6.0  10.0
  SPEED  0.0  400.  500.  400.  0.
```

SPEE OMEGA1 OMEGA2 .... OMEGAN

Status - REQUIRED

Full Keyword - SPEED

Function - Defines the speed of rotation of the part.

Input Variables -

OMEGA1,OMEGA2,...,OMEGAN (Real) -REQUIRED

Speed of rotation (RPM) at times specified on TIME card.

Additional Information -

This card may be input as often as required. Each card begins with the keyword SPEE.

Examples of Use -

1. Request an input for three-dimensional centrifugal case

```
**BODY FORCE
  CENT
  DIRE   0.0  0.0  1.0
  POINT  0.0  0.0  0.0
  TIME   1.0  2.0  3.0
  SPEED  100.  200.  400.
```

5.6.3 INERTIAL BODY FORCE

INER

Status - OPTIONAL

Full Keyword - INERTIA FORCE

Function - Indicates that an inertia force will be applied.

Input Variables - NONE

Additional Information -

Only one (time dependent) inertia load condition may be defined for an analysis. It is applied to the entire body..

Examples of Use -

1. Request inertial input

```
**BODY FORCE
  INER
  DIRE   1.0  0.0  0.0
  TIME   1.0
  ACCE   300.
```

DIRE X Y Z

Status - OPTIONAL

Full Keyword - `DIRECTION`

Function - Defines a vector parallel to the direction of inertia force.

Input Variables -

X,Y,Z (Real) - REQUIRED

Cartesian components of a vector parallel to the inertia force.

Additional Information -

Only one direction can be defined in an analysis.

If this card is omitted, the inertia force is assumed to be parallel to the z-axis of the global system in the negative direction (i.e. gravity loading).

Examples of Use -

1. Defines an inertial force in the positive Z-direction for a three-dimensional analysis.

```
**BODY FORCE
 INER
 DIRE  0.  0.  1.
 TIME  1.0
 ACCE  10.0
```

TIME T1 T2 ... TN

Status - REQUIRED (if INER is input)

Full Keyword - TIMES

Function - Defines the times at which the acceleration of the body will be defined.

Input Variables -

T1,T2,...,TN (Real) - REQUIRED

Times at which acceleration will be defined.

Additional Information -

This card may be input as many times as required. Each card begins with the keyword TIME.

A maximum of 20 time values may be specified.

Examples of Use -

1. Specifies accelerations at three times.

```
**BODY FORCE
  INER
  DIRE  0.    0.    1.
  TIME  1.0   2.0   3.0
  ACCE  10.0  15.0  20.0
```

ACCE ACC1 ACC2 .... ACCN

Status - OPTIONAL

Full Keyword - ACCELERATION

Function - Defines the acceleration of the body.

Input Variables -

ACC1,ACC2,...,ACCN (Real) -REQUIRED

Acceleration at times specified on TIME card.

Additional Information -

This card may be input as often as required. Each card begins with the keyword ACCE.

DEFAULT: Gravity loading of 386.4 in/sec/sec.

Examples of Use -

1. Specifies an acceleration of 100.0 units in an inertial body force loading at a single time step.

```
**BODY FORCE
  INER
  DIRE  0.   1.
  TIME  1.
  ACCE  100.
```

5.6.4 THERMAL BODY FORCE

THER

Status - OPTIONAL

Full Keyword - THERMAL

Function - Indicates the start of temperature input for thermal body force.

Input Variables - NONE

Additional Information -

All thermal input for an analysis should be input in a single **BODY FORCE data block, even though more than one GMR may be involved.

This flag is not relevant to heat transfer analysis.

Examples of Use -

1. Thermal input for a two region problem at three time points.

```
**BODY FORCE
  THERM
  TIME  2.0  4.0  6.0
  GMR  REG1
  TEMP
   1    100.0    200.0    500.0
   6    100.0    200.0    550.0
   5    100.0    200.0    525.0
          .        .        .
          .        .        .
  27    100.0    200.0    475.0
  GMR  REG2
  TEMP
  40    100.0    300.0    500.0
  41    100.0    300.0    475.0
          .        .        .
          .        .        .
```

TIME T1 T2 .... TN

Status - REQUIRED (if THER is input)

Full Keyword - TIMES

Function - Defines the times at which the nodal point temperatures will be defined.

Input Variables -

T1,T2,...,TN (Real) - REQUIRED

Times at which nodal point temperatures will be defined.

Additional Information -

If all times do not fit on one card, TIME may be continued on a second card immediately following the first time card, starting with the keyword TIME. Only one time definition is allowed for thermal input, and therefore temperatures in each GMR should be defined according to this one definition.

A maximum of 20 time values may be specified.

Examples of Use -

1. Specifies three times at which temperatures are input.

```
**BODY FORCE
  THERM
  TIME    1.0     2.0     3.0
  GMR  GMR1
  TEMP
  1       100.0   200.0   400.0
  2       150.0   300.0   600.0
  3       200.0   400.0   800.0
```

GMR IDGMR

Status - REQUIRED (if THER is input)

Full Keyword - GMR

Function - Identifies the GMR in which temperatures will be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometric definition (NAME on ID card in **GMR input.)

Additional Information -

The GMR card is repeated for each GMR. All GMR's (for which thermal input is desired) should be contained under one **BODY FORCE input.

Examples of Use -

1. Identifies GMR name at which temperatures are to be specified.

```
**BODY FORCE
  THERM
  TIME    1.0
  GMR     REG1
```

TEMP

Status - REQUIRED (if THER is input)

Full Keyword - TEMPERATURES

Function - Signals the beginning of input for this GMR of temperatures at nodal points as a function of time.

Input Variables - None.

Additional Information - None.

Examples of Use -

1. Defines temperature distribution at previously defined nodes.

```
**BODY FORCE
  THERM
  TIME  1.0
  GMR   GMR1
  TEMP
  101    1.0
  103    2.0
  105   25.0
  107   30.0
  109   35.0
```

(NONE) `NNODE TEM1 TEM2 ... TEMN`

Status - REQUIRED (if THER is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Defines nodal point temperatures at times specified on TIME card.

Input Variables -

NNODE (Integer) - REQUIRED

User nodal point number for which temperatures are being specified on this card.

TEM1,TEM2,...,TEMN (Real) - REQUIRED

The temperatures at node NNODE at the times specified on the TIME card.

Additional Information -

This card is input as many times as required for each nodal point. Each new card begins with the nodal point number.

Temperature input for each nodal point must begin on a new card.

Examples of Use -

1. Defines discrete temperature distribution at previously defined nodes.

```
**BODY FORCE
  THERM
  TIME    1.0
  GMR     GMR1
  TEMP
  101     10.0
  103     20.0
  105     25.0
  107     30.0
  109     35.0
```

5.6.5 EMBEDDED HOT SPOTS

HOT

Status - OPTIONAL

Full Keyword - `HOT SPOT`

Function - Signals that input will be provided defining the temperature history at buried hot spots.

Input Variables - NONE

Additional Information -

The times at which hot spot temperatures will be specified and the temperature histories are defined using TIME and TEMP cards.

Examples of Use -

1. Hot spot input for two regions with two hot spots per region (two time points).

```
**BODY FORCE
  HOT SPOT
  TIME  1.0  10.0
  GMR  REG1
  TEMP
      1001   0.1    100.0    1000.0
      2001   0.5    100.0    1500.0
  GMR  REG2
  TEMP
      4005   0.5    100.0    3000.0
      4006   1.2    100.0    1500.0
```

TIME T1 T2 .... TN

Status - REQUIRED (if HOT is input)

Full Keyword - TIMES

Function - Defines the times at which hot spot temperatures will be defined.

Input Variables -

T1,T2,...,TN (Real) - REQUIRED

Times at which hot spot temperatures will be defined.

Additional Information -

If all times do not fit on one card, Times may be continued on a second card immediately following the first time card, starting with the keyword TIME. Only one time definition is allowed for thermal input, and therefore temperatures in each GMR should be defined according to this one definition.

A maximum of 20 time values may be specified.

Examples of Use -

1. Defines time 1.0 and 3.0 at which temperatures are input.

```
**BODY FORCE
  HOT SPOT
  TIME   1.0   3.0
  GMR GMR1
  TEMP
    1001   0.2   100.0   1000.0
    2001   0.4   100.0   1500.0
  GMR GMR2
  TEMP
    4005   0.1   100.0   3000.0
    4006   0.3   100.0   1500.0
```

GMR IDGMR

Status - REQUIRED (if HOT is input)

Full Keyword - GMR

Function - Identifies the GMR in which temperatures will be defined.

Input Variables -

IDGMR (Alphanumeric) - REQUIRED

IDGMR is the identifier for the GMR as input during the geometric definition (NAME on ID card in **GMR input.)

Additional Information -

The GMR card is repeated for each GMR. All GMR's (for which thermal input is desired) should be contained under one **BODY FORCE input.

Examples of Use -

1. Defines GMR at which temperatures are input.

```
**BODY FORCE
 HOT SPOT
 TIME  1.0
 GMR   REG1
 TEMP
 101   0.1   150.0
 105   0.1   120.0
 108   0.2   135.0
 111   0.3   142.0
```

TEMP

Status - REQUIRED (if HOT is input)

Full Keyword - TEMPERATURES

Function - Signals the beginning of input of temperatures at hot spots as a function of time.

Input Variables - NONE

Additional Information -

This card is followed by data cards defining hot spot temperatures. Each hot spot is characterized by its centroid and volume.

(NONE) NNODE VOL TEM1 TEM2 ... TEMN

Status - REQUIRED (if HOT is input)

Full Keyword - NO KEYWORD REQUIRED

Function - Defines hot spot temperatures at times specified on TIME card.

Input Variables -

NNODE (Integer) - REQUIRED
User nodal point number of the hot spot centroid.

VOL (Real) - REQUIRED
Volume of hot spot.

TEM1,TEM2,...,TEMN (Real) - REQUIRED
The temperatures at node NNODE at the times specified on the TIME card.

Additional Information -

This card is input as many times as required for each hot spot. Each new card begins with the nodal point number.

Temperature input for each hot spot must begin on a new card.

A maximum of 10 hot spots are permitted per GMR.

Examples of Use -

1. Defines the list of temperature entries.

```
**BODY FORCE
  HOT SPOT
  TIME  1.0
```

```
GMR   GMR1
101   0.1    150.0
105   0.1    120.0
108   0.2    135.0
111   0.3    142.0
```

5.7 DEFINITION OF BOUNDARY CONDITION CHANGES

BEST3D allows for a change of an essential boundary condition to a natural boundary condition and vice-versa during a single analysis. This means that a boundary condition for a component of a particular node may be specified at the outset of the analysis, and then be changed to an unknown boundary condition during the course of the run. Similarly an unknown boundary condition may be changed during an analysis to a specified boundary condition and then changed back again to an unknown, as many times as neccessary. This is useful, for instance, in a plastic metal forming analysis when a body is loaded in one direction and then changed to a loading in the other direction. Similarly, in a heat transfer analysis, a body can be initially insulated on a side and then changed later in the run to allow a temperature flow across that same face.

Essentially, the first definition of the boundary conditions for the initial time, is input in the usual manner. This first definition may be time dependent and may be used for any number of solution times defined in the case input. An entire new definition of boundary conditions (with changes in the specified knowns and unknowns) are input, preceded by the **BCCH and TIME card, after the initial definition.

Included on the TIME card is the time at which the changeover takes place. This next set likewise be time-dependent. This procedure may be repeated as many times as neccessary.

SECTION	KEYWORD	PURPOSE
5.7.1 Changes in Boundary Condition Knowns and Unknowns		
	**BCCH	start of the new B.C. definition
	TIME	defines the time the change will occur

5.7.1 BOUNDARY CONDITION CHANGE CARD

**BCCH

Status - OPTIONAL (Not available for all analysis types)

Full Keyword - **BCCHANGE

Function - Identifies an entirely new boundary condition definition in which the specified known and unknown boundary conditions will differ from the previously specified set.

Input Variables - NONE

Additional Information -

A TIME card must immediately follow the **BCCH card.

New boundary conditions for the entire body must be input cooresponding to each **BCCH card. Even if certain **BCSETS do not change during the entire analysis, they must be repeated (redefined) for each **BCCH card.

All boundary condition sets preceding the **BCCH card are forgotten once the time of solution has passed over to the next set of boundary conditions after the **BCCH card.

Examples of Use -

1. Analysis with three unique sets of known and unknown boundary conditions (**BCCH card is used twice)

```
            ALL THE BCSETS
            **BCSET11
            **BCSET12
                  .
                  .
            **BCSET1n




**BCCHANGE
  TIME  10.0
             ALL THE BCSETS AFTER FIRST CHANGE
             **BCSET21
             **BCSET22
                  .
                  .
             **BCSET2n
```

```
**BCCHANGE
  TIME  20.0
         ALL THE BCSETS AFTER SECOND CHANGES
        **BCSET31
        **BCSET32
              .
              .
        **BCSET3n
```

As a particular case :

```
**BCSET
  ID U1FIX
  VALUE
  LOCAL
  GMR REG2
    SURFACE SIDE1
    DISP 1
    SPLIST ALL
    T 1 0.0



**BCCH
  TIME  10.0

**BCSET
  ID U1FIX
  VALUE
  LOCAL
  GMR REG2
    SURFACE SIDE1
    TRAC 1
    SPLIST ALL
    T 1 0.0



**BCCH
  TIME  20.0

**BCSET
  ID U1FIX
  VALUE
  LOCAL
  GMR REG2
    SURFACE SIDE1
```

```
DISP 1
SPLIST ALL
T 1 0.0
```

TIME VALUE

Status - REQUIRED (if **BCCH is input)

Full Keyword - TIME

Function - Identifies the time at which the boundary condition change will occur.

Input Variables -

VALUE (Real) - REQUIRED

The time value at which the change occurs

Additional Information -

A new boundary condition definition (**BCSETS) must be given for the entire body immediately following the TIME card.

The change occurs for the first solution time (specified in the case input) greater than the VALUE specified on the TIME card of the **BCCH and less than the VALUE specified on the TIME cards of the subsequent **BCCH changes.

If more than one **BCCH card is present, the VALUE of time on the subsequent cards should be in ascending order.

The times specified on the TIME card within a BCSET may unneccessarily extend over the range allowed by the TIME card of the **BCCH change. In any case, the TIME card for **BCCH has priority.

Examples of Use -

1. Specifies the time at which the boundary condition change will occur.

```
**BCCH
  TIME  10.0

**BCSET
  ID  U1FIX
  VALUE
  LOCAL
    .
    .
    .
```

6.0 EXAMPLE PROBLEMS

In this chapter example problems are presented to illustrate data preparation for **BEST3D**. An attempt has been made to keep the problem geometry as simple as possible so the user is not burdened with undue complexity. It is hoped that an analyst who is using an analysis procedure for the first time will find these example problems invaluable in the learning process.

The example problems are divided into the following categories based on **BEST3D** analysis type:

ELASTICITY
FORCED-VIBRATION
FREE-VIBRATION
HEAT TRANSFER
PLASTICITY
TRANSIENT DYNAMICS
STRESS ANALYSIS WITH HOLES AND INSERTS

Each problem includes the following items:

1) A Brief Problem Description
2) Geometry and Boundary Element Model
3) Input Data for running the problem in **BEST3D**
4) Selected Output from **BEST3D**

It should be noted that since the boundary element models illustrated utilize coarse meshes, the **BEST3D** results may differ somewhat from the theoretical values. However, with a finer mesh, the theoretical values should be obtained. Also, the results may vary somewhat depending on the computer system being used to run **BEST3D**.

An estimated RUN TIME is cited for each problem to give the user a feeling for the computer time needed to run the problem. All RUN TIMES are related to problem ELAS605, a simple elastic cube in tension, which will be considered to have a run time of 1 unit. A different problem which has a RUN TIME of 8 would take approximately eight times longer to run that the ELAS605 problem. However, these times will vary somewhat depending on the computer system being used to run **BEST3D**.

EXAMPLE PROBLEM: ELAS602

ANALYSIS TYPE:
THERMALELASTICITY,
3-D, STATIC, ELASTIC ANALYSIS WITH THERMAL BODY FORCE

PROBLEM DESCRIPTION:
TWO REGION, FREE EXPANSION OF A RECTANGULAR CUBE SUBJECTED TO A UNIFORM AND A LINEAR TEMPERATURE DISTRIBUTION

BOUNDARY ELEMENT MODEL:
TWO SQUARE REGIONS, WITH SIX QUADRATIC BOUNDARY ELEMENTS PER REGION. VOLUME INTEGRATION DISCRETIZATION APPROACH.

REFERENCE FOR ANALYTICAL SOLUTION:
BOLEY AND WEINNER (1960), THEORY OF THERMAL STRESSES.
SOLUTION FOR FREE EXPANSION OF A UNRESTRAINED BODY UNDER UNIFORM AND LINEAR TEMPERATURE DISTRIBUTIONS

SOLUTION POINTS TO VERIFY:

TIME 1: UNIFORM TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE LINEAR, AND STRESSES ARE ZERO (<<100, E*ALPHA*T)

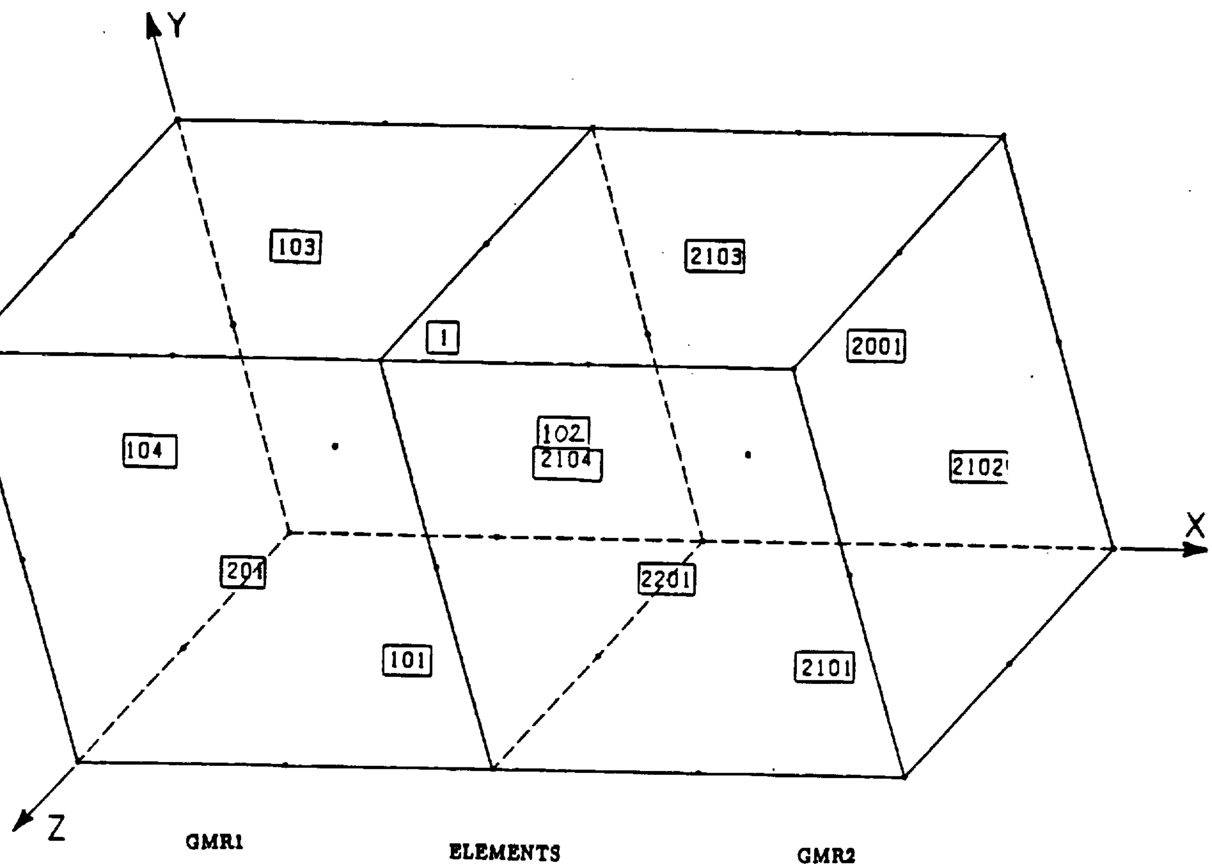
	X-DISPLACEMENT		XX-STRESS	
NODE	ANAL.	BEST	ANAL.	BEST
8007	1.0	.99998	.0	.0004
8009	2.0	1.99998	.0	-.0005

TIME 2: LINEAR TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE QUADRATIC AND STRESSES ARE ZERO (<<100, E*ALPHA*T)

	X-DISPLACEMENT		XX-STRESS	
NODE	ANAL.	BEST	ANAL.	BEST
8007	0.5	.499998	.0	.0025
8009	2.0	2.000000	.0	-.0025

RUN TIME:
7 X BASE PROBLEM

MISCELLANEOUS:
STRESSES ARE ZERO RELATIVE TO (E*ALPHA*T).
THIS EXAMPLE CAN ALSO BE RUN USING THE GLOBAL SHAPE FUNCTION APPROACH.


Y
X
Z
103
2103
1
2001
104
102
2104
2102
204
2201
101
2101
GMR1
ELEMENTS
GMR2

```
**CASE CONTROL
  TITLE  FREE THERMAL EXPANSION OF A 3-D RECTANGULAR CUBE
  TIME  1.0  2.0


**MATERIAL INPUT
  ID MAT1
  TEMP 70.
  EMOD 100.
  POIS 0.3
  ALPHA  1.


$
$  DEFINE GEOMETRY OF REGION 1
$
**GMR
  ID GMR1
  MAT MAT1
  TREF 70.                 $ TEMPERATURE AT WHICH MATERIAL CONSTANTS
                           $  ARE EVALUATED FOR INTEGRATION
  TINI 0.0                 $ CHANGE IN TEMPERATURE DATUM
  POINTS
        1       .0000         .0000         .0000
        2       .5000         .0000         .0000
        3      1.0000         .0000         .0000
        4      1.0000         .5000         .0000
        5      1.0000        1.0000         .0000
        6       .5000        1.0000         .0000
        7       .0000        1.0000         .0000
        8       .0000         .5000         .0000
     1001       .0000         .0000         .5000
     1003      1.0000         .0000         .5000
     1005      1.0000        1.0000         .5000
     1007       .0000        1.0000         .5000
     2001       .0000         .0000        1.0000
     2002       .5000         .0000        1.0000
     2003      1.0000         .0000        1.0000
     2004      1.0000         .5000        1.0000
     2005      1.0000        1.0000        1.0000
     2006       .5000        1.0000        1.0000
     2007       .0000        1.0000        1.0000
     2008       .0000         .5000        1.0000
     4001       .5            .5            .5
  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
     101        1     2     3  1003  2003  2002  2001  1001
     102        3     4     5  1005  2005  2004  2003  1003
     103        5     6     7  1007  2007  2006  2005  1005
     104        7     8     1  1001  2001  2008  2007  1007
       1        1     2     3     4     5     6     7     8
     201     2001  2002  2003  2004  2005  2006  2007  2008
    NORMAL  201 +
```

```
      VOLUME
        TYPE QUAD
        CELLS
     1      1      2      3      4      5      6      7      8   1001   1003
     1   1005   1007   2001   2002   2003   2004   2005   2006   2007   2008
4001
      FULL
      SAMPLING POINTS
        4007   1.0  .5   .5


    $
    $  DEFINE GEOMETRY OF REGION 2
    $
    **GMR
      ID GMR2
      MAT MAT1
      TREF 70.             $ TEMPERATURE AT WHICH MATERIAL CONSTANTS
                           $  ARE EVALUATED FOR INTEGRATION
      TINI 0.0             $ CHANGE IN TEMPERATURE DATUM
      POINTS
         5001       1.0000        .0000        .0000
         5002       1.5000        .0000        .0000
         5003       2.0000        .0000        .0000
         5004       2.0000        .5000        .0000
         5005       2.0000       1.0000        .0000
         5006       1.5000       1.0000        .0000
         5007       1.0000       1.0000        .0000
         5008       1.0000        .5000        .0000
         6001       1.0000        .0000        .5000
         6003       2.0000        .0000        .5000
         6005       2.0000       1.0000        .5000
         6007       1.0000       1.0000        .5000
         7001       1.0000        .0000       1.0000
         7002       1.5000        .0000       1.0000
         7003       2.0000        .0000       1.0000
         7004       2.0000        .5000       1.0000
         7005       2.0000       1.0000       1.0000
         7006       1.5000       1.0000       1.0000
         7007       1.0000       1.0000       1.0000
         7008       1.0000        .5000       1.0000
         8001       1.5           .5           .5
      SURFACE SURF21
        TYPE QUAD
        ELEMENTS
        2101     5001  5002  5003  6003  7003  7002  7001  6001
        2102     5003  5004  5005  6005  7005  7004  7003  6003
        2103     5005  5006  5007  6007  7007  7006  7005  6005
        2104     5007  5008  5001  6001  7001  7008  7007  6007
        2001     5001  5002  5003  5004  5005  5006  5007  5008
        2201     7001  7002  7003  7004  7005  7006  7007  7008
        NORMAL  2201 +
      VOLUME
        TYPE QUAD
        CELLS
     2   5001   5002   5003   5004   5005   5006   5007   5008   6001   6003
     2   6005   6007   7001   7002   7003   7004   7005   7006   7007   7008
8001
```

```
 FULL
 SAMPLING POINTS
   8002     1.25  .25  .25
   8007     1.0   .5   .5
   8009     2.0   .5   .5


$
$  DEFINE INTERFACE CONNECTION
$
**INTERFACE
  GMR GMR1
  SURFACE SURF11
  ELEMENT  102
  GMR GMR2
  SURFACE SURF21
  ELEMENT  2104


$
$  FIX BODY TO PREVENT RIGID BODY DISPLACEMENT BUT ALLOW FREE EXPANSION
$
**BCSET
  ID  BC1
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   104
    POINT 1
  DISP  1
    SPLIST 1
      T  1   0.0
  DISP  2
    SPLIST 1
      T  1   0.0
  DISP  3
    SPLIST 1
      T  1   0.0

**BCSET
  ID  BC2
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   104
    POINTS    7
  DISP  3
    SPLIST  7
      T  1   0.0

**BCSET
  ID  BC3
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   104
    POINTS    2001
  DISP  1
    SPLIST  2001
      T  1  0.0
```

```
**BCSET
  ID  BC4
  GMR  GMR2
  SURFACE  SURF21
    ELEMENTS   2101
    POINTS   5003
  DISP  2
    SPLIST  5003
      T  1  0.0


$
$  DEFINE TEMPERATURE DISTRIBUTION FOR ALL VOLUME SOURCE POINTS
$    (NODAL POINTS OF THE VOLUME CELL)
$
**BODY FORCE
  THERMAL
  TIME 1. 2.
   GMR GMR1
   TEMP
        1  1.       .0000
        2  1.       .5000
        3  1.      1.0000
        4  1.      1.0000
        5  1.      1.0000
        6  1.       .5000
        7  1.       .0000
        8  1.       .0000
     1001  1.       .0000
     1003  1.      1.0000
     1005  1.      1.0000
     1007  1.       .0000
     2001  1.       .0000
     2002  1.       .5000
     2003  1.      1.0000
     2004  1.      1.0000
     2005  1.      1.0000
     2006  1.       .5000
     2007  1.       .0000
     2008  1.       .0000
     4001  1.       .5
   GMR GMR2
   TEMP
     5001  1.      1.0000
     5002  1.      1.5000
     5003  1.      2.0000
     5004  1.      2.0000
     5005  1.      2.0000
     5006  1.      1.5000
     5007  1.      1.0000
     5008  1.      1.0000
     6001  1.      1.0000
     6003  1.      2.0000
     6005  1.      2.0000
     6007  1.      1.0000
     7001  1.      1.0000
     7002  1.      1.5000
     7003  1.      2.0000
```

```
        7005  1.       2.0000
        7006  1.       1.5000
        7007  1.       1.0000
        7008  1.       1.0000
        7004  1.       2.0000
        8001  1.       1.5
$
$   END OF DATA
```

JOB TITLE: FREE THERMAL EXPANSION OF A 3-D RECTANGULAR CUBE (PLANE STRESS)
LOAD CALCULATION AT TIME = 1.000000

LOADS FOR REGION GMR1

ELEMENT	X	Y	Z
101	0.00000E+00	0.00000E+00	0.00000E+00
102	-0.29030E-05	-0.58303E-05	0.20256E-04
103	0.00000E+00	0.00000E+00	0.00000E+00
104	0.15131E-05	0.13253E-05	-0.13164E-04
1	0.00000E+00	0.00000E+00	0.00000E+00
201	0.00000E+00	0.00000E+00	0.00000E+00
LOAD BALANCE	-0.13899E-05	-0.45050E-05	0.70918E-05

LOADS FOR REGION GMR2

ELEMENT	X	Y	Z
2101	0.00000E+00	-0.85136E-05	0.00000E+00
2102	0.00000E+00	0.00000E+00	0.00000E+00
2103	0.00000E+00	0.00000E+00	0.00000E+00
2104	0.29030E-05	0.58303E-05	-0.20256E-04
2001	0.00000E+00	0.00000E+00	0.00000E+00
2201	0.00000E+00	0.00000E+00	0.00000E+00
LOAD BALANCE	0.29030E-05	-0.26833E-05	-0.20256E-04

JOB TITLE: FREE THERMAL EXPANSION OF A 3-D RECTANGULAR CUBE (PLANE STRESS)
INTERIOR STRESS AT TIME = 1.0000 FOR REGION = GMR2

NODE	SIGMA-XX	SIGMA-YY	SIGMA-ZZ	TAU-XY	TAU-XZ	TAU-YZ
5001	0.112942E-03	-0.135469E-04	0.269794E-04	0.299227E-05	0.329729E-04	0.217919E-04
5002	0.224097E-04	-0.207505E-04	0.318042E-04	0.137731E-04	0.643175E-05	0.000000E+00
5003	-0.323922E-04	-0.997077E-04	0.618313E-04	0.230653E-04	-0.208005E-04	0.204505E-04
5004	-0.135993E-04	-0.554654E-04	0.505251E-04	0.144001E-04	0.000000E+00	0.798500E-05
5005	0.404480E-04	-0.922881E-05	0.545673E-04	-0.172706E-05	-0.202122E-04	-0.159941E-04
5006	0.243403E-05	0.303978E-05	0.420816E-04	-0.520531E-05	-0.868853E-05	0.000000E+00
5007	-0.432490E-04	-0.186673E-04	0.148589E-04	-0.258991E-05	0.600672E-05	-0.133627E-04
5008	0.167745E-04	-0.383891E-04	0.183250E-04	0.169018E-04	-0.899209E-05	0.122502E-04
6001	-0.290777E-04	-0.149196E-05	-0.484691E-04	-0.441010E-05	0.946793E-04	0.203372E-04
6003	0.113100E-04	-0.110123E-04	-0.856356E-05	0.000000E+00	-0.199230E-04	0.220608E-04
6005	0.276191E-04	-0.204476E-04	-0.286016E-04	0.000000E+00	-0.129472E-04	-0.317163E-04
6007	-0.686579E-04	-0.337796E-04	-0.398314E-04	-0.805605E-05	0.371088E-04	-0.186852E-04
7001	-0.179568E-03	0.514656E-04	-0.814144E-04	0.631960E-05	0.110604E-04	0.371597E-05
7002	-0.451465E-04	0.302081E-04	-0.719804E-04	-0.125817E-04	0.664831E-06	0.000000E+00
7003	0.685535E-04	0.504503E-04	-0.852095E-04	-0.325843E-04	-0.109650E-04	0.959433E-05
7004	0.536639E-04	0.325078E-04	-0.671512E-04	-0.487026E-05	0.000000E+00	-0.742931E-05
7005	0.883981E-04	-0.331587E-05	-0.852593E-04	0.299630E-04	-0.400474E-07	-0.231471E-04

7006	-0.182534E-04	-0.341219E-04	-0.376223E-04	0.146717E-04	0.116924E-04	0.000000E+00
7007	-0.140968E-03	-0.474573E-04	-0.650144E-04	-0.827937E-05	0.276376E-04	-0.106161E-04
7008	-0.114938E-03	0.183998E-04	-0.299056E-04	-0.108488E-04	-0.161112E-04	-0.115414E-05
8001	-0.313929E-04	-0.235588E-04	-0.225379E-04	0.364919E-06	0.135010E-04	0.289346E-05
8002	-0.190755E-05	-0.211490E-04	-0.875142E-05	0.598322E-05	0.316275E-04	0.977089E-05
8007	-0.352388E-04	-0.205789E-04	-0.237930E-04	-0.281723E-05	0.339229E-04	0.599176E-05
8009	-0.175492E-05	-0.117582E-04	-0.133781E-04	0.806576E-07	-0.343067E-05	-0.226579E-05

EXAMPLE PROBLEM: ELAS605

ANALYSIS TYPE: 3-D LINEAR ELASTICITY

PROBLEM DESCRIPTION:
SIMPLE CUBE IN TENSION IN X-DIRECTION.

BOUNDARY ELEMENT MODEL:
SINGLE CUBE COMPOSED OF SIX BOUNDARY ELEMENTS.
SAMPLING POINTS INCLUDED TO MONITOR INTERIOR BEHAVIOR.

REFERENCE FOR ANALYTICAL SOLUTION:
CRANDALL AND DAHL, AN INTRODUCTION TO THE MECHANICS OF SOLIDS, SECOND EDITION (1972), PG. 81-84. X-DISPLACEMENT = FL/AE.

SOLUTION POINTS TO VERIFY:

(X-DISPLACEMENT)

NODE	ANALYTICAL	BEST
12	0.0001	0.0001

(Y-DISPLACEMENT)

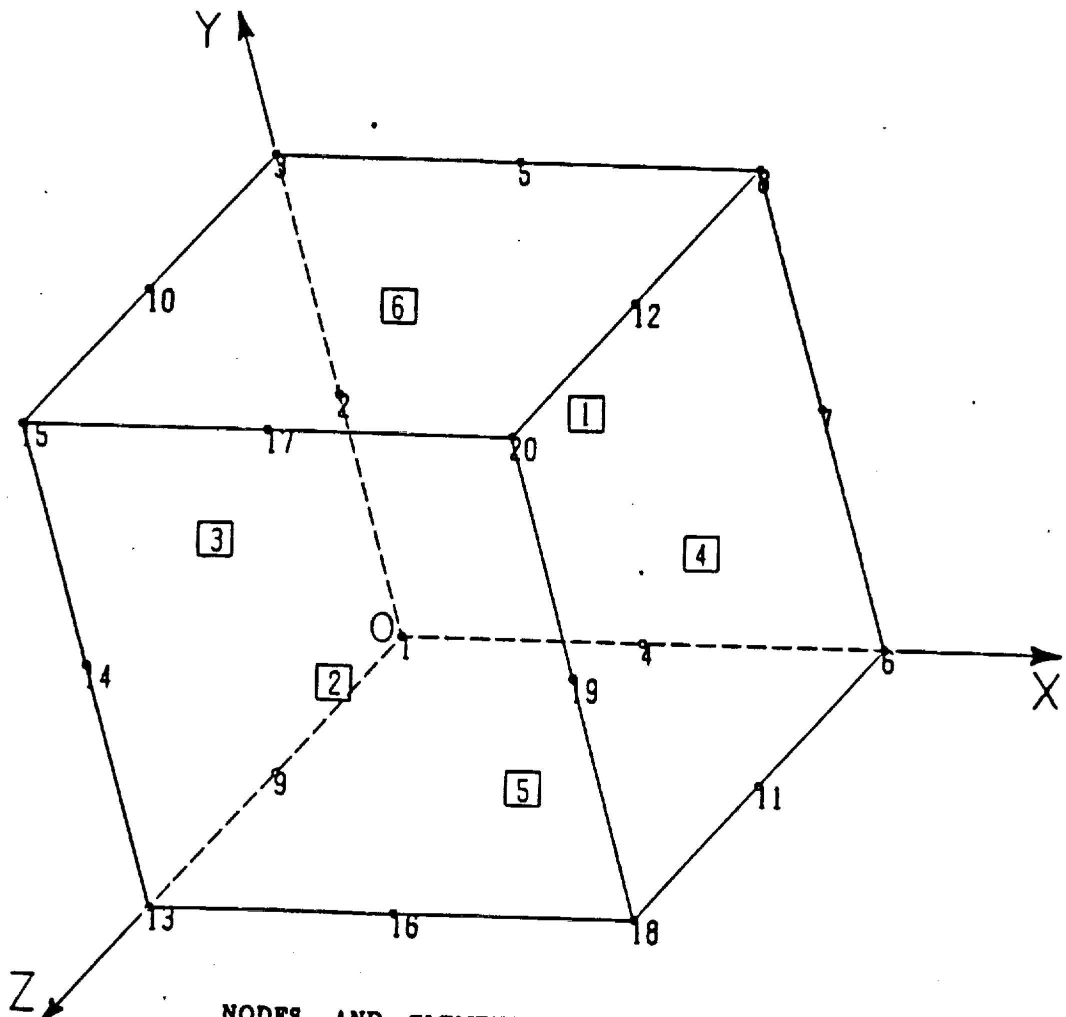
NODE	ANALYTICAL	BEST
12	-0.00003000	-0.00002999

(X-STRESS)

NODE	ANALYTICAL	BEST
12	1000.	999.9

RUN TIME:
1.0 X BASE PROBLEM (THIS IS THE BASE PROBLEM!)

MISCELLANEOUS:



NODES AND ELEMENTS

```
**CASE
  TITLE     A UNIT CUBE IN SIMPLE TENSION
  ELASTICITY


**MATE
  ID MAT1
  TEMP  70.0
    EMOD 10.E6
    POIS 0.3


**GMR
  ID GMR1
  MAT MAT1
  TREF  70.0
  POINTS
   1         0.0       0.0       0.0
   2         0.0       0.5       0.0
   3         0.0       1.0       0.0
   4         0.5       0.0       0.0
   5         0.5       1.0       0.0
   6         1.0       0.0       0.0
   7         1.0       0.5       0.0
   8         1.0       1.0       0.0
   9         0.0       0.0       0.5
  10         0.0       1.0       0.5
  11         1.0       0.0       0.5
  12         1.0       1.0       0.5
  13         0.0       0.0       1.0
  14         0.0       0.5       1.0
  15         0.0       1.0       1.0
  16         0.5       0.0       1.0
  17         0.5       1.0       1.0
  18         1.0       0.0       1.0
  19         1.0       0.5       1.0
  20         1.0       1.0       1.0
  SURFACE SURF1
    TYPE QUAD
    ELEMENTS
  1     1  2  3  5  8  7  6  4
  2    13 14 15 17 20 19 18 16
  3     1  2  3 10 15 14 13  9
  4     6  7  8 12 20 19 18 11
  5     1  4  6 11 18 16 13  9
  6     3  5  8 12 20 17 15 10
  NORMAL  1 +
  SAMPLING POINTS
       21       0.5     0.0     0.5
       22       0.5     0.25    0.5
       23       0.5     0.5     0.5
       24       0.5     0.75    0.5
       25       0.5     1.0     0.5
```

```
**BCSET
  ID DISP1
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 3
  DISP 1
    SPLIST ALL
    T 1       0.0

**BCSET
  ID DISP2
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 5
  DISP 2
    SPLIST ALL
    T 1       0.0

**BCSET
  ID DISP3
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 1
  DISP 3
    SPLIST ALL
    T 1       0.0

**BCSET
  ID TRAC1
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 4
  TRAC 1
    SPLIST ALL
    T 1       1000.0

$ END OF DATA
```

```
**** CASE CONTROL INPUT ****

JOB TITLE       A UNIT CUBE IN SIMPLE TENSION


   PLANE STRESS  FLAG:    0            PLANE STRAIN     FLAG:    0
   AXISYMMETRY   FLAG:    0            DIMENSIONALITY   FLAG:    3
   PLASTICITY    FLAG:    0            THERMAL          FLAG:    0
   CENTRIFUGAL   FLAG:    0            INERTIAL LOAD    FLAG:    0
   COUPLED       FLAG:    0            COUPLED THERMAL  FLAG:    0
   CONSOLIDATION FLAG:    0            DIFFUSION        FLAG:    0
   BOUNDARY RESTART  :    0            GLOBAL SHAPE FUNCTION:    0
   DOMAIN RESTART    :    0            STEADY-STATE  FLAG   :    0

   THERMAL  INHOMOGENEITY   FLAG:    0

   MATERIAL INHOMOGENEITY   FLAG:    0

   NON-ITERATIVE PLASTICITY FLAG:    0
   TRANSIENT DYNAMICS       FLAG:    0
   STEADY STATE DYNAMIC     FLAG:    0
   FLUID DYNAMICS           FLAG:    0

   NUMBER OF DEGREES OF  FREEDOM:    3

   BOUNDARY INTEGRATION  EPSILON:   0.00100000
   INTERIOR INTEGRATION  EPSILON:   0.00100000

   PRINTING FLAGS FOR BOUNDARY VALUES :
      BOUNDARY DISP. AND TRAC.              1
      NODAL DISP. ,STRESS AND STRAIN        1
      LOAD CALCULATION                      1


     **** MATERIAL INPUT ****

   MATERIAL NAME: MAT1

   ELASTIC
   ISOTROPIC MATERIAL

      DENSITY:           0.0000E+00
      DAMPING COEFF:     0.0000
      POISSONS RATIO:    0.3000


        TEMP          ALPHA            E
     0.70000E+02   0.00000E+00    0.10000E+08


     **** GMR INPUT ****
```

```
REGION     1
NAME GMR1               MATERIAL MAT1

REFERENCE TEMPERATURE            70.00000
INITIAL TEMPERATURE OF GMR        0.00000

NODES    25             ELEMENTS     6        SURFACES     1
SOURCE POINTS    20     CELLS     0           INFINITE ELEMENTS    0
ADDITIONAL JUMP TERM  0.000000                ENCLOSING ELEMENTS    0
GLOBAL SHAPE FUNCTION NODES     0             HOLE ELEMENTS     0
NUMBER OF INSERTS     0                       INSERT ELEMENTS     0


INFORMATIONAL MESSAGE  1001
   NOT ALL POINTS ARE DEFINED AS BOUNDARY SOURCE POINTS
```

COORDINATE LIST

NODE	X	Y	Z
1	0.00000	0.00000	0.00000
2	0.00000	0.50000	0.00000
3	0.00000	1.00000	0.00000
4	0.50000	0.00000	0.00000
5	0.50000	1.00000	0.00000
6	1.00000	0.00000	0.00000
7	1.00000	0.50000	0.00000
8	1.00000	1.00000	0.00000
9	0.00000	0.00000	0.50000
10	0.00000	1.00000	0.50000
11	1.00000	0.00000	0.50000
12	1.00000	1.00000	0.50000
13	0.00000	0.00000	1.00000
14	0.00000	0.50000	1.00000
15	0.00000	1.00000	1.00000
16	0.50000	0.00000	1.00000
17	0.50000	1.00000	1.00000
18	1.00000	0.00000	1.00000
19	1.00000	0.50000	1.00000
20	1.00000	1.00000	1.00000
21	0.50000	0.00000	0.50000
22	0.50000	0.25000	0.50000
23	0.50000	0.50000	0.50000
24	0.50000	0.75000	0.50000
25	0.50000	1.00000	0.50000

SURFACE SURF1 QUADRATIC VARIATION

ELEMENT	NODES							
1	1	2	3	5	8	7	6	4
2	13	16	18	19	20	17	15	14
3	1	9	13	14	15	10	3	2
4	6	7	8	12	20	19	18	11
5	1	4	6	11	18	16	13	9
6	3	10	15	17	20	12	8	5

SOURCE POINT LIST

```
                    1    2    3    4    5    6    7    8    9   10   11   12   13   14   15   16   17
   18   19   20


          **** BOUNDARY CONDITION INPUT ****

      BOUNDARY CONDITION SET NAME DISP1          TYPE: VALUE
      GMR GMR1            SURFACE SURF1

      ELEMENT LIST
          3

      SOURCE POINT LIST
          1     9    13    14    15    10     3     2

      COMPONENT   1   DISPLACEMENT INPUT

      DATA VALUES:
       0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00


          **** BOUNDARY CONDITION INPUT ****

      BOUNDARY CONDITION SET NAME DISP2          TYPE: VALUE
      GMR GMR1            SURFACE SURF1

      ELEMENT LIST
          5

      SOURCE POINT LIST
          1     4     6    11    18    16    13     9

      COMPONENT   2   DISPLACEMENT INPUT

      DATA VALUES:
       0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00


          **** BOUNDARY CONDITION INPUT ****

      BOUNDARY CONDITION SET NAME DISP3          TYPE: VALUE
      GMR GMR1            SURFACE SURF1

      ELEMENT LIST
          1

      SOURCE POINT LIST
          1     2     3     5     8     7     6     4

      COMPONENT   3   DISPLACEMENT INPUT

      DATA VALUES:
       0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00  0.00000E+00
```

**** BOUNDARY CONDITION INPUT ****

BOUNDARY CONDITION SET NAME TRAC1 TYPE: VALUE
GMR GMR1 SURFACE SURF1

ELEMENT LIST
4

SOURCE POINT LIST
6 7 8 12 20 19 18 11

COMPONENT 1 TRACTION INPUT

DATA VALUES:
0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04 0.10000E+04

** ERROR SUMMARY AFTER INPUT PHASE **

FATAL ERRORS: 0
WARNING MESSAGES: 0
INFORMATIONAL MESSAGES: 1

INFORMATIONAL MESSAGES LISTED ABOVE:
1001

1

BEGIN SURFACE INTEGRATION OF GMR GMR1

HIGH PRECISION

SOURCE POINTS

INTERIOR POINTS

END SURFACE INTEGRATION OF GMR GMR1

MATRIX DECOMPOSITION - DIAGONAL BLOCK 1
CONDITION NUMBER 0.16294E+03

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
BOUNDARY SOLUTION AT TIME = 0.0000 FOR REGION = GMR1

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	Z-DISPL.	X-TRAC.	Y TRAC.	Z TRAC.
1	1	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.22270E+00
1	2	0.00000E+00	-0.15005E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.26818E+00
1	3	0.00000E+00	-0.30004E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.10535E+00
1	5	0.50004E-04	-0.30011E-04	0.00000E+00	0.00000E+00	0.00000E+00	-0.58124E-01
1	8	0.99994E-04	-0.29991E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.20986E+00
1	7	0.10001E-03	-0.15000E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.20890E+00
1	6	0.10001E-03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.36048E+00
1	4	0.50009E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	-0.16577E+00

2	13	0.00000E+00	0.00000E+00	-0.30004E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	16	0.50004E-04	0.00000E+00	-0.30011E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	18	0.99994E-04	0.00000E+00	-0.29991E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	19	0.10001E-03	-0.15000E-04	-0.29997E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	20	0.99987E-04	-0.29994E-04	-0.29994E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	17	0.50000E-04	-0.30010E-04	-0.30010E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	15	0.00000E+00	-0.30005E-04	-0.30005E-04	0.00000E+00	0.00000E+00	0.00000E+00
2	14	0.00000E+00	-0.15005E-04	-0.30005E-04	0.00000E+00	0.00000E+00	0.00000E+00
3	1	0.00000E+00	0.00000E+00	0.00000E+00	-0.10005E+04	0.00000E+00	0.00000E+00
3	9	0.00000E+00	0.00000E+00	-0.15005E-04	-0.10001E+04	0.00000E+00	0.00000E+00
3	13	0.00000E+00	0.00000E+00	-0.30004E-04	-0.10005E+04	0.00000E+00	0.00000E+00
3	14	0.00000E+00	-0.15005E-04	-0.30005E-04	-0.10002E+04	0.00000E+00	0.00000E+00
3	15	0.00000E+00	-0.30005E-04	-0.30005E-04	-0.10004E+04	0.00000E+00	0.00000E+00
3	10	0.00000E+00	-0.30005E-04	-0.15005E-04	-0.10002E+04	0.00000E+00	0.00000E+00
3	3	0.00000E+00	-0.30004E-04	0.00000E+00	-0.10005E+04	0.00000E+00	0.00000E+00
3	2	0.00000E+00	-0.15005E-04	0.00000E+00	-0.10001E+04	0.00000E+00	0.00000E+00
4	6	0.10001E-03	0.00000E+00	0.00000E+00	0.10000E+04	0.00000E+00	0.00000E+00
4	7	0.10001E-03	-0.15000E-04	0.00000E+00	0.10000E+04	0.00000E+00	0.00000E+00
4	8	0.99994E-04	-0.29991E-04	0.00000E+00	0.10000E+04	0.00000E+00	0.00000E+00
4	12	0.10001E-03	-0.29997E-04	-0.15000E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	20	0.99987E-04	-0.29994E-04	-0.29994E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	19	0.10001E-03	-0.15000E-04	-0.29997E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	18	0.99994E-04	0.00000E+00	-0.29991E-04	0.10000E+04	0.00000E+00	0.00000E+00
4	11	0.10001E-03	0.00000E+00	-0.15000E-04	0.10000E+04	0.00000E+00	0.00000E+00
5	1	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.22275E+00	0.00000E+00
5	4	0.50009E-04	0.00000E+00	0.00000E+00	0.00000E+00	-0.16570E+00	0.00000E+00
5	6	0.10001E-03	0.00000E+00	0.00000E+00	0.00000E+00	0.36057E+00	0.00000E+00
5	11	0.10001E-03	0.00000E+00	-0.15000E-04	0.00000E+00	0.20893E+00	0.00000E+00
5	18	0.99994E-04	0.00000E+00	-0.29991E-04	0.00000E+00	0.21049E+00	0.00000E+00
5	16	0.50004E-04	0.00000E+00	-0.30011E-04	0.00000E+00	-0.58155E-01	0.00000E+00
5	13	0.00000E+00	0.00000E+00	-0.30004E-04	0.00000E+00	0.10541E+00	0.00000E+00
5	9	0.00000E+00	0.00000E+00	-0.15005E-04	0.00000E+00	0.26828E+00	0.00000E+00

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
BOUNDARY SOLUTION AT TIME = 0.0000 FOR REGION = GMR1

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	Z-DISPL.	X-TRAC.	Y TRAC.	Z TRAC.
6	3	0.00000E+00	-0.30004E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
6	10	0.00000E+00	-0.30005E-04	-0.15005E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	15	0.00000E+00	-0.30005E-04	-0.30005E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	17	0.50000E-04	-0.30010E-04	-0.30010E-04	0.00000E+00	0.00000E+00	0.00000E+00

6	20	0.99987E-04	-0.29994E-04	-0.29994E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	12	0.10001E-03	-0.29997E-04	-0.15000E-04	0.00000E+00	0.00000E+00	0.00000E+00
6	8	0.99994E-04	-0.29991E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
6	5	0.50004E-04	-0.30011E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
LOAD CALCULATION AT TIME = 0.000000

LOADS FOR REGION GMR1

ELEMENT	X	Y	Z
1	0.00000E+00	0.00000E+00	0.95296E-02
2	0.00000E+00	0.00000E+00	0.00000E+00
3	-0.10000E+04	0.00000E+00	0.00000E+00
4	0.10000E+04	0.00000E+00	0.00000E+00
5	0.00000E+00	0.95156E-02	0.00000E+00
6	0.00000E+00	0.00000E+00	0.00000E+00
LOAD BALANCE	-0.20203E-01	0.95156E-02	0.95296E-02

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
NODAL OUTPUT AT TIME = 0.000000 FOR REGION = GMR1

NODE	DISPLACEMENT	--------- STRESS ---------		--------- STRAIN ---------	
	X/Y/Z	XX/YY/ZZ	XY/XZ/YZ	XX/YY/ZZ	XY/XZ/YZ
1	0.00000E+00	0.10003E+04	0.00000E+00	0.10004E-03	0.00000E+00
	0.00000E+00	-0.12547E+00	0.00000E+00	-0.30017E-04	0.00000E+00
	0.00000E+00	-0.12546E+00	0.00000E+00	-0.30017E-04	0.00000E+00
2	0.00000E+00	0.10002E+04	-0.42452E-01	0.10003E-03	-0.55188E-08
	-0.15005E-04	-0.36594E-01	0.00000E+00	-0.30004E-04	0.00000E+00
	0.00000E+00	-0.19430E+00	-0.96197E-04	-0.30025E-04	-0.12506E-10
3	0.00000E+00	0.10003E+04	-0.53161E-01	0.10003E-03	-0.69109E-08
	-0.30004E-04	0.11022E+00	0.00000E+00	-0.29997E-04	0.00000E+00
	0.00000E+00	-0.35271E-01	-0.13619E-02	-0.30016E-04	-0.17705E-09
4	0.50009E-04	0.10001E+04	0.46846E-01	0.10001E-03	0.60900E-08
	0.00000E+00	-0.66880E-02	0.46853E-01	-0.30002E-04	0.60909E-08
	0.00000E+00	-0.66681E-02	0.00000E+00	-0.30002E-04	0.00000E+00
5	0.50004E-04	0.99992E+03	-0.41151E-01	0.99994E-04	-0.53496E-08
	-0.30011E-04	0.13870E-01	0.49553E-01	-0.29994E-04	0.64420E-08
	0.00000E+00	-0.90094E-01	0.00000E+00	-0.30007E-04	0.00000E+00
6	0.10001E-03	0.99975E+03	0.60773E-01	0.99990E-04	0.79005E-08
	0.00000E+00	-0.25114E+00	0.60773E-01	-0.30010E-04	0.79005E-08
	0.00000E+00	-0.25113E+00	0.00000E+00	-0.30010E-04	0.00000E+00

7	0.10001E-03	0.99991E+03	0.40438E-01	0.99995E-04	0.52569E-08
	-0.15000E-04	0.17284E-01	0.00000E+00	-0.29991E-04	0.00000E+00
	0.00000E+00	-0.14902E+00	-0.16546E-01	-0.30013E-04	-0.21510E-08
8	0.99994E-04	0.99976E+03	-0.33954E-02	0.99976E-04	-0.44141E-09
	-0.29991E-04	0.10892E+00	0.66612E-01	-0.29978E-04	0.86596E-08
	0.00000E+00	-0.12321E+00	-0.24540E-01	-0.30008E-04	-0.31902E-08
9	0.00000E+00	0.10002E+04	0.00000E+00	0.10003E-03	0.00000E+00
	0.00000E+00	-0.19431E+00	-0.42451E-01	-0.30025E-04	-0.55186E-08
	-0.15005E-04	-0.36621E-01	-0.10669E-03	-0.30004E-04	-0.13870E-10
10	0.00000E+00	0.10002E+04	0.00000E+00	0.10002E-03	0.00000E+00
	-0.30005E-04	0.52551E-01	-0.29686E-01	-0.30003E-04	-0.38592E-08
	-0.15005E-04	0.36850E-01	-0.20376E-02	-0.30005E-04	-0.26489E-09
11	0.10001E-03	0.99991E+03	0.00000E+00	0.99995E-04	0.00000E+00
	0.00000E+00	-0.14889E+00	0.40439E-01	-0.30013E-04	0.52571E-08
	-0.15000E-04	0.17349E-01	-0.16598E-01	-0.29991E-04	-0.21578E-08
12	0.10001E-03	0.99990E+03	0.00000E+00	0.99985E-04	0.00000E+00
	-0.29997E-04	0.10396E+00	0.32747E-01	-0.29988E-04	0.42571E-08
	-0.15000E-04	0.59250E-01	0.89620E-02	-0.29994E-04	0.11651E-08
13	0.00000E+00	0.10003E+04	0.00000E+00	0.10003E-03	0.00000E+00
	0.00000E+00	-0.35310E-01	-0.53161E-01	-0.30016E-04	-0.69109E-08
	-0.30004E-04	0.11018E+00	-0.13642E-02	-0.29997E-04	-0.17735E-09

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
NODAL OUTPUT AT TIME = 0.000000 FOR REGION = GNR1

NODE	DISPLACEMENT X/Y/Z	STRESS XX/YY/ZZ	STRESS XY/XZ/YZ	STRAIN XX/YY/ZZ	STRAIN XY/XZ/YZ
14	0.00000E+00	0.10002E+04	-0.29636E-01	0.10002E-03	-0.38526E-08
	-0.15005E-04	0.36823E-01	0.00000E+00	-0.30005E-04	0.00000E+00
	-0.30005E-04	0.52612E-01	-0.20376E-02	-0.30003E-04	-0.26489E-09
15	0.00000E+00	0.10002E+04	-0.37327E-01	0.10002E-03	-0.48525E-08
	-0.30005E-04	0.10503E+00	-0.37366E-01	-0.30000E-04	-0.48576E-08
	-0.30005E-04	0.10514E+00	-0.25862E-02	-0.30000E-04	-0.33621E-09
16	0.50004E-04	0.99992E+03	0.49539E-01	0.99994E-04	0.64401E-08
	0.00000E+00	-0.89941E-01	-0.41141E-01	-0.30007E-04	-0.53483E-08
	-0.30011E-04	0.13901E-01	0.00000E+00	-0.29994E-04	0.00000E+00
17	0.50000E-04	0.99986E+03	-0.43995E-01	0.99987E-04	-0.57194E-08
	-0.30010E-04	-0.18402E-01	-0.43992E-01	-0.29997E-04	-0.57189E-08
	-0.30010E-04	-0.18250E-01	0.00000E+00	-0.29997E-04	0.00000E+00
18	0.99994E-04	0.99976E+03	0.66575E-01	0.99976E-04	0.86548E-08
	0.00000E+00	-0.12329E+00	-0.33861E-02	-0.30008E-04	-0.44020E-09

	-0.29991E-04	0.10891E+00	-0.24587E-01	-0.29978E-04	-0.31963E-08
19	0.10001E-03	0.99990E+03	0.32693E-01	0.99985E-04	0.42501E-08
	-0.15000E-04	0.58983E-01	0.00000E+00	-0.29994E-04	0.00000E+00
	-0.29997E-04	0.10406E+00	0.89480E-02	-0.29988E-04	0.11632E-08
20	0.99987E-04	0.99976E+03	-0.20816E-01	0.99969E-04	-0.27061E-08
	-0.29994E-04	0.10793E+00	-0.20753E-01	-0.29985E-04	-0.26979E-08
	-0.29994E-04	0.10841E+00	0.34034E-01	-0.29985E-04	0.44244E-08

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
INTERIOR DISPLACEMENT AT TIME = 0.0000 FOR REGION = GNR1

NODE	X DISPLACEMENT	Y DISPLACEMENT	Z DISPLACEMENT
21	0.500135E-04	0.000000E+00	-0.150089E-04
22	0.500090E-04	-0.750441E-05	-0.150050E-04
23	0.500105E-04	-0.150052E-04	-0.150052E-04
24	0.500067E-04	-0.225076E-04	-0.150046E-04
25	0.500091E-04	-0.300125E-04	-0.150074E-04

JOB TITLE: A UNIT CUBE IN SIMPLE TENSION
INTERIOR STRESS AT TIME = 0.0000 FOR REGION = GNR1

NODE	SIGMA-XX	SIGMA-YY	SIGMA-ZZ	TAU-XY	TAU-XZ	TAU-YZ
21	0.100017E+04	0.981304E-01	-0.332382E-01	0.000000E+00	0.546809E-03	0.000000E+00
22	0.999768E+03	0.227315E+00	-0.385985E-01	-0.165191E+00	-0.125456E-02	0.643814E-01
23	0.100012E+04	-0.556403E-01	-0.556958E-01	-0.231984E-02	-0.232729E-02	0.584437E-02
24	0.999633E+03	0.393963E+00	-0.154854E+00	0.160049E+00	-0.247152E-03	-0.514198E-01
25	0.100003E+04	0.000000E+00	-0.882590E-01	0.000000E+00	0.225028E-02	0.000000E+00

END OF ANALYSIS

VERIFICATION PROBLEM: D.402

ANALYSIS TYPE:
THERMOELASTICITY,
3-D, STATIC, ELASTIC ANALYSIS WITH A THERMAL HOT SPOT

PROBLEM DISCRIPTION:
FREE EXPANSION OF A RECTANGULAR CUBE SUBJECTED TO A THERMAL HOT SPOT OF UNIT VOLUME

BOUNDARY ELEMENT MODEL:
OCTAL SYMMETRY MODEL CONSISTING OF TWO SQUARE REGIONS, WITH THREE QUADRATIC BOUNDARY ELEMENTS IN REGION ONE AND FOUR IN REGION TWO.

REFERENCE FOR ANALYTICAL SOLUTION:
BOLEY AND WEINNER (1960), THEORY OF THERMAL STRESSES.
SOLUTION FOR FREE EXPANSION OF A UNRESTRAINED BODY UNDER UNIFORM AND LINEAR TEMPERATURE DISTRIBUTIONS

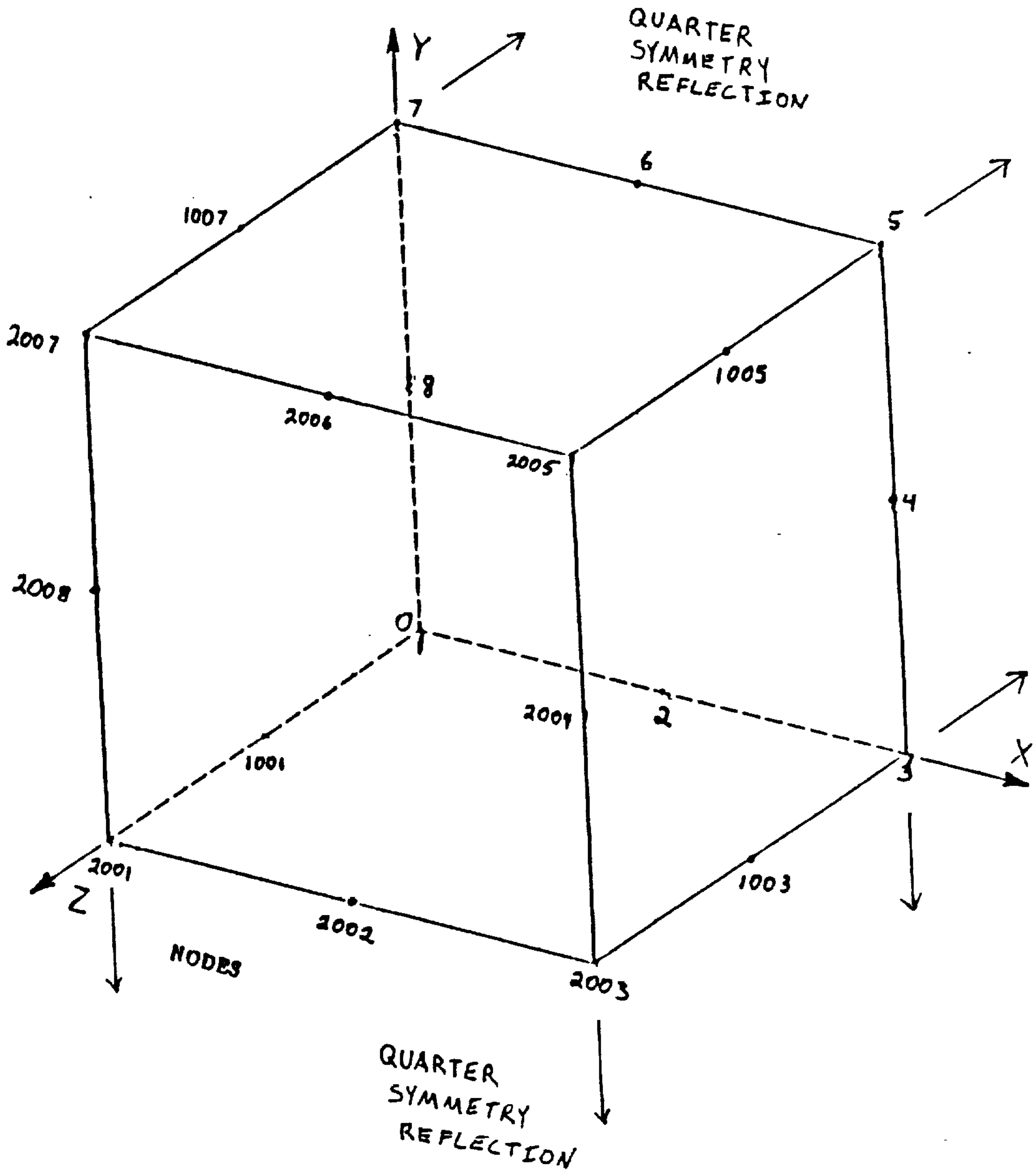
SOLUTION POINTS TO VERIFY:

TIME 1: UNIFORM TEMPERATURE DISTRIBUTION-DISPLACEMENTS ARE LINEAR, AND STRESSES ARE ZERO (<<100, E*ALPHA*T)

GMR	NODE	X-DISP.	Y-DISP.	XX-STRESS
GMR1	3000	.25000	.25000	.0010
GMR2	3005	.74997	.24998	-.0044

RUN TIME:
1.3 X BASE PROBLEM

MISCELLANEOUS:
STRESSES ARE ZERO RELETIVE TO (E*ALPHA*T).


QUARTER SYMMETRY REFLECTION
Y
7
6
5
1007
2007
1005
8
2006
2005
4
2008
0
1
2004
2
X
1001
3
2001
Z
1003
2002
NODES
2003
QUARTER SYMMETRY REFLECTION

```
**CASE
  TITLE  - D.402 -  3-D 2-REGION HOTSPOT
    TIMES  1.
    SYMM OCTA
    ECHO

**MATE
  ID MAT1
  TEMP 0.0
  EMOD  100.
  POIS  0.3
  ALPHA 1.

 **GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
        1        .0000       .0000       .0000
        2        .5000       .0000       .0000
        3       1.0000       .0000       .0000
        4       1.0000       .5000       .0000
        5       1.0000      1.0000       .0000
        6        .5000      1.0000       .0000
        7        .0000      1.0000       .0000
        8        .0000       .5000       .0000
     1001        .0000       .0000       .5000
     1003       1.0000       .0000       .5000
     1005       1.0000      1.0000       .5000
     1007        .0000      1.0000       .5000
     2001        .0000       .0000      1.0000
     2002        .5000       .0000      1.0000
     2003       1.0000       .0000      1.0000
     2004       1.0000       .5000      1.0000
     2005       1.0000      1.0000      1.0000
     2006        .5000      1.0000      1.0000
     2007        .0000      1.0000      1.0000
     2008        .0000       .5000      1.0000
 SURFACE SURF11
   TYPE QUAD
   ELEMENTS
    102       3     4     5  1005  2005  2004  2003  1003
    103       5     6     7  1007  2007  2006  2005  1005
    201    2001  2002  2003  2004  2005  2006  2007  2008
   NORMAL  201 +
   SAMPLING POINT
   3000   .5 .5 .5

**GMR
  ID GMR2
  MAT MAT1
  TREF 70.0
  POINTS
        1       1.0000       .0000       .0000
```

```
            2         1.5000         .0000         .0000
            3         2.0000         .0000         .0000
            4         2.0000         .5000         .0000
            5         2.0000        1.0000         .0000
            6         1.5000        1.0000         .0000
            7         1.0000        1.0000         .0000
            8         1.0000         .5000         .0000
         1001         1.0000         .0000         .5000
         1003         2.0000         .0000         .5000
         1005         2.0000        1.0000         .5000
         1007         1.0000        1.0000         .5000
         2001         1.0000         .0000        1.0000
         2002         1.5000         .0000        1.0000
         2003         2.0000         .0000        1.0000
         2004         2.0000         .5000        1.0000
         2005         2.0000        1.0000        1.0000
         2006         1.5000        1.0000        1.0000
         2007         1.0000        1.0000        1.0000
         2008         1.0000         .5000        1.0000
   SURFACE SURF21
     TYPE QUAD
     ELEMENTS
      102        3     4     5  1005  2005  2004  2003  1003
      103        5     6     7  1007  2007  2006  2005  1005
      104        7     8     1  1001  2001  2008  2007  1007
      201     2001  2002  2003  2004  2005  2006  2007  2008
     NORMAL  201 +
     SAMPLING POINT
     3005  1.5 .5 .5

**INTERFACE
   GMR GMR1
    SURFACE SURF11
      ELEMENT 102
   GMR GMR2
    SURFACE SURF21
      ELEMENT 104

$   INPUT FOR HOT SPOT

**BODY FORCE
  HOT SPOT
  TIME 1.  3.
  GMR GMR1
   TEMP
     3000 1. 0.5  1.5
  GMR GMR2
   TEMP
     3005 1. 0.5  1.5

$ END OF DATA
```

EXAMPLE PROBLEM: FORC601

ANALYSIS TYPE: DYNAMIC ANALYSIS
3-D, STEADY-STATE

PROBLEM DESCRIPTION:
DYNAMIC STIFFNESS OF RIGID SURFACE SQUARE FOOTING RESTING ON ELASTIC HALF-SPACE (SOIL).

BOUNDARY ELEMENT MODEL:
BECAUSE OF QUADRATIC SYMMETRY IN GEOMATRY AND LOAD, ONLY ONE QUARTER OF THE GEOMETRY IS MODELLED. 4 ELEMENTS ARE USED TO DISCRETIZE THE SOIL-FOUNDATION INTERFACE WHILE THE FREE-SURFACE IS MODELLED WITH 6 ELEMENTS.

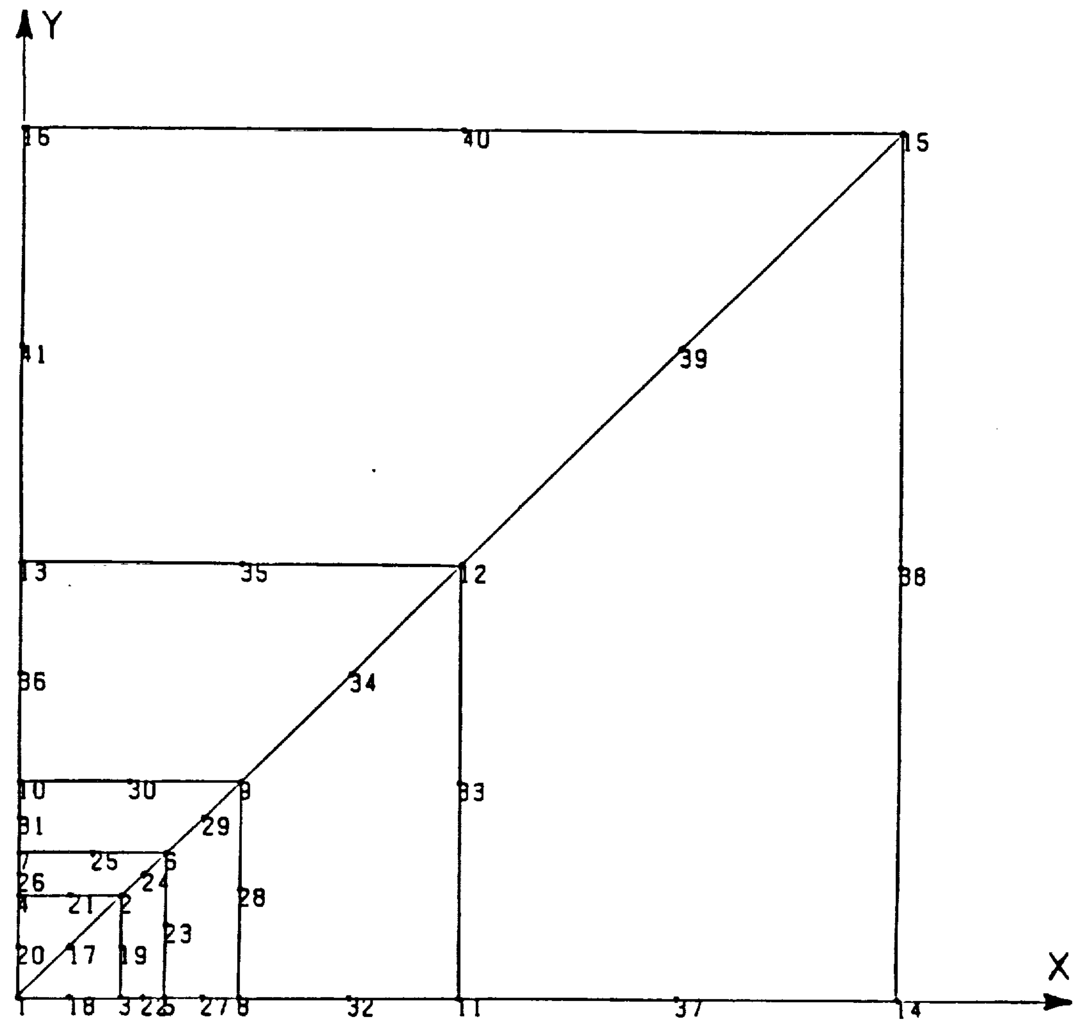
THE FOUNDATION IS SUBJECTED TO UNIT VERTICAL DISPLACEMENT (DIRECTION 3) WHILE TRACTIONS ON THE OTHER TWO DIRECTIONS ARE ASSUMED TO BE ZERO. THIS CORRESPONDS TO THE SO-CALLED 'RELAXED' CONDITION. THE FREE-SURFACE IS ASSUMED TO BE TRACTION FREE.

SOLUTION POINTS TO VERIFY:

EXCITING CIRCULAR FREQUENCY	ELEMENT	VERTICAL LOAD (BEST) (DIRECTION 3) REAL PART	COMPLEX PART
0.001	1	0.3553E+00	0.2878E-03
0.001	3	0.5434E+00	0.4398E-03
1.5	1	0.1242E+00	0.5871E+00
1.5	3	0.4411E+00	0.8158E+00

RUN TIME:
20 X BASE PROBLEM

MISCELLANEOUS:


Y
X
16
40
15
41
39
13
35
12
38
36
34
10
30
9
33
31
29
7
25
6
26
24
4
21
2
28
23
20
17
19
1
18
3
22
5
27
8
32
11
37
14

NODES

```
**CASE CONTROL
  TITLE     SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)
   SYMMETRY QUAD
   FORCED   .001   1.5
   PRINT LOAD


**MATE
    ID  MAT1
    TEMP 70.
    EMOD 2.66666
    POIS 0.33333
    DENS 1.0
    DAMP 0.0


**GMR
    ID GMR1
    MAT MAT1
    TREF 70.
    HALF            $  SINCE THE PROBLEM INVOLVES HALF-SPACE,
                    $  THE CARD 'HALF' IS USED IN THE GMR INPUT.
                    $  THIS ELIMINATES THE NEED FOR 'ENCLOSING ELEMENTS'.
    POINTS
         1       0.000000       0.000000       0.000000
         2       0.700000       0.700000       0.000000
         3       0.700000       0.000000       0.000000
         4       0.000000       0.700000       0.000000
         5       1.000000       0.000000       0.000000
         6       1.000000       1.000000       0.000000
         7       0.000000       1.000000       0.000000
         8       1.500000       0.000000       0.000000
         9       1.500000       1.500000       0.000000
        10       0.000000       1.500000       0.000000
        11       3.000000       0.000000       0.000000
        12       3.000000       3.000000       0.000000
        13       0.000000       3.000000       0.000000
        14       6.000000       0.000000       0.000000
        15       6.000000       6.000000       0.000000
        16       0.000000       6.000000       0.000000
        17       0.350000       0.350000       0.000000
        18       0.350000       0.000000       0.000000
        19       0.700000       0.350000       0.000000
        20       0.000000       0.350000       0.000000
        21       0.350000       0.700000       0.000000
        22       0.850000       0.000000       0.000000
        23       1.000000       0.500000       0.000000
        24       0.850000       0.850000       0.000000
        25       0.500000       1.000000       0.000000
        26       0.000000       0.850000       0.000000
        27       1.250000       0.000000       0.000000
        28       1.500000       0.750000       0.000000
        29       1.250000       1.250000       0.000000
        30       0.750000       1.500000       0.000000
```

```
        31     0.000000     1.250000     0.000000
        32     2.250000     0.000000     0.000000
        33     3.000000     1.500000     0.000000
        34     2.250000     2.250000     0.000000
        35     1.500000     3.000000     0.000000
        36     0.000000     2.250000     0.000000
        37     4.500000     0.000000     0.000000
        38     6.000000     3.000000     0.000000
        39     4.500000     4.500000     0.000000
        40     3.000000     6.000000     0.000000
        41     0.000000     4.500000     0.000000
  SURFACE SURF
    TYPE QUAD
    ELEMENTS
           1     3    19     2    17     1    18
           2     2    21     4    20     1    17
           3     3    22     5    23     6    24     2    19
           4     2    24     6    25     7    26     4    21
           5     5    27     8    28     9    29     6    23
           6     6    29     9    30    10    31     7    25
           7     8    32    11    33    12    34     9    28
           8     9    34    12    35    13    36    10    30
           9    11    37    14    38    15    39    12    33
          10    12    39    15    40    16    41    13    35
   NORMAL  1 +

**BCSET
   ID DISP3
    VALUE
    GMR GMR1
    SURFACE SURF
    ELEMENTS   1  2  3  4
    DISP  3
    SPLIST  ALL
    T 1    1.
$
$ END OF DATA
```

JOB TITLE: SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)
LOAD CALCULATION AT EXCITING FREQUENCY = 0.001000

LOADS FOR REGION GMR1

ELEMENT	X		Y		Z	
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.3553E+00	0.2878E-03
2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.3553E+00	0.2878E-03
3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.5434E+00	0.4398E-03
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.5434E+00	0.4398E-03
5	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
6	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
7	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
8	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
9	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
10	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
LOAD BALANCE	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1797E+01	0.1455E-02

1

JOB TITLE: SQUARE FOOTING ON HALF SPACE (10 ELEM, 41 NODES)
LOAD CALCULATION AT EXCITING FREQUENCY = 1.500000

LOADS FOR REGION GMR1

ELEMENT	X		Y		Z	
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1242E+00	0.5871E+00
2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1242E+00	0.5871E+00
3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4411E+00	0.8158E+00
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.4411E+00	0.8159E+00
5	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
6	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
7	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
8	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
9	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
10	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
LOAD BALANCE	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1131E+01	0.2806E+01

C-4

EXAMPLE PROBLEM: FREE601

ANALYSIS TYPE: 3-D FREE VIBRATION

PROBLEM DESCRIPTION:
DETERMINATION OF THE NATURAL FREQUENCY OF A SQUARE CANTILEVER (1*1*6.5), YOUNG'S MODULUS=10000, DENSITY=1.0, POISSON'S RATIO=0.

BOUNDARY ELEMENT MODEL:
TEN 8-NODED QUADRATIC SURFACE ELEMENTS, ONE ON EACH END AND TWO ON EACH SIDE. GLOBAL BOUNDARY CONDITIONS SPECIFIED EVERYWHERE.

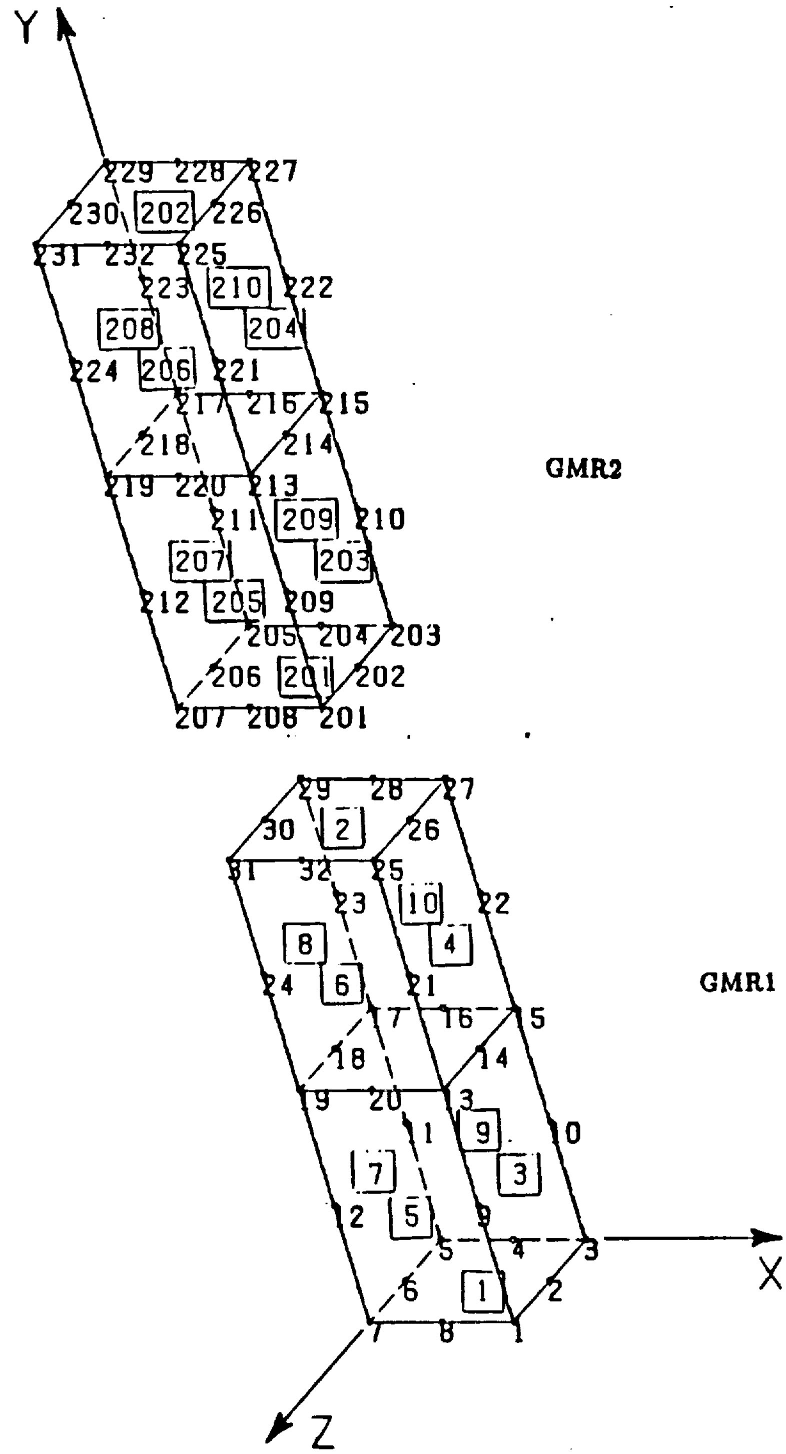
REFERENCE FOR COMPARED SOLUTION:
S. AHMAD AND P.K. BANERJEE, FREE VIBRATION ANALYSIS BY BEM USING PARTICULAR INTEGRALS, JOURNAL OF ENGINEERING MECHANICS, ASCE, VOL. 112, NO. 7, 682-695, (1986).

SOLUTION POINTS TO VERIFY:

MODE	FREQUENCY BY PRESENT MESH	BEST(HZ) FINE MESH	FEM RESULT(HZ) (BY MARC-HOST)
1	.39055	0.368	0.378
2	.39055	0.368	0.378
3	2.3424	2.214	2.188

RUN TIME:
13 X BASE PROBLEM

MISCELLANEOUS:


GMR2
GMR1
Y
X
Z

```
**CASE
  TITLE  FREE VIBRATION OF A 3-D CANTILEVER(1*1*6.5) BY 2 REGION MESH.
  FREE 4


**MATE
  ID MAT1
    TEMP      70.
    POIS     0.000001
    EMOD     10000.
    DENSITY  1.0


**GMR
  ID GMR1
  MAT MAT1
  POINTS
        1  1.0  0.0  1.0
        2  1.0  0.0  0.5
        3  1.0  0.0  0.0
        4  0.5  0.0  0.0
        5  0.0  0.0  0.0
        6  0.0  0.0  0.5
        7  0.0  0.0  1.0
        8  0.5  0.0  1.0
        9  1.0  0.8125  1.0
       10  1.0  0.8125  0.0
       11  0.0  0.8125  0.0
       12  0.0  0.8125  1.0
       13  1.0  1.625 1.0
       14  1.0  1.625 0.5
       15  1.0  1.625 0.0
       16  0.5  1.625 0.0
       17  0.0  1.625 0.0
       18  0.0  1.625 0.5
       19  0.0  1.625 1.0
       20  0.5  1.625 1.0
       21  1.0  2.4375  1.0
       22  1.0  2.4375   0.0
       23  0.0  2.4375   0.0
       24  0.0  2.4375   1.0
       25  1.0  3.25  1.0
       26  1.0  3.25  0.5
       27  1.0  3.25  0.0
       28  0.5  3.25  0.0
       29  0.0  3.25  0.0
       30  0.0  3.25  0.5
       31  0.0  3.25  1.0
       32  0.5  3.25  1.0
  SURFACE SURF1
    TYPE QUAD
    ELEMENTS
    1     1  2  3  4  5  6  7  8
    2    25 26 27 28 29 30 31 32
```

```
  3    1  2  3 10 15 14 13  9
  4   13 14 15 22 27 26 25 21
  5    7  8  1  9 13 20 19 12
  6   19 20 13 21 25 32 31 24
  7    7  6  5 11 17 18 19 12
  8   19 18 17 23 29 30 31 24
  9    5  4  3 10 15 16 17 11
 10   17 16 15 22 27 28 29 23
NORMAL  1 -


**GMR
 ID GMR2
 MAT MAT1
 POINTS
        201  1.0  3.25   1.0
        202  1.0  3.25   0.5
        203  1.0  3.25   0.0
        204  0.5  3.25   0.0
        205  0.0  3.25   0.0
        206  0.0  3.25   0.5
        207  0.0  3.25   1.0
        208  0.5  3.25   1.0
        209  1.0  4.0625   1.0
        210  1.0  4.0625   0.0
        211  0.0  4.0625   0.0
        212  0.0  4.0625   1.0
        213  1.0  4.875 1.0
        214  1.0  4.875 0.5
        215  1.0  4.875 0.0
        216  0.5  4.875 0.0
        217  0.0  4.875 0.0
        218  0.0  4.875 0.5
        219  0.0  4.875 1.0
        220  0.5  4.875 1.0
        221  1.0  5.6875   1.0
        222  1.0  5.6875   0.0
        223  0.0  5.6875   0.0
        224  0.0  5.6875   1.0
        225  1.0  6.5   1.0
        226  1.0  6.5   0.5
        227  1.0  6.5   0.0
        228  0.5  6.5   0.0
        229  0.0  6.5   0.0
        230  0.0  6.5   0.5
        231  0.0  6.5   1.0
        232  0.5  6.5   1.0
 SURFACE SURF1
   TYPE QUAD
   ELEMENTS
   201     201  202  203  204  205  206  207  208
   202     225  226  227  228  229  230  231  232
   203     201  202  203  210  215  214  213  209
   204     213  214  215  222  227  226  225  221
   205     207  208  201  209  213  220  219  212
   206     219  220  213  221  225  232  231  224
   207     207  206  205  211  217  218  219  212
   208     219  218  217  223  229  230  231  224
```

```
    209     205  204  203  210  215  216  217  211
    210     217  216  215  222  227  228  229  223
  NORMAL   201 -

**INTERFACE
$ INTERFACE CONDITION
$ FIRST REGION
  GMR GMR1
  SURFACE SURF1
     ELEMENT 2
$ SECOND REGION
  GMR GMR2
  SURFACE SURF1
     ELEMENT 201

**BCSET
  ID  BCS1
  VALUE
  GMR GMR1
    SURFACE  SURF1
      ELEMENT   1
        DISP 1
        SPLIST ALL
        T 1 0.0
        DISP 2
        SPLIST ALL
        T 1 0.0
        DISP 3
        SPLIST ALL
        T 1 0.0
$
$  END OF DATA
```

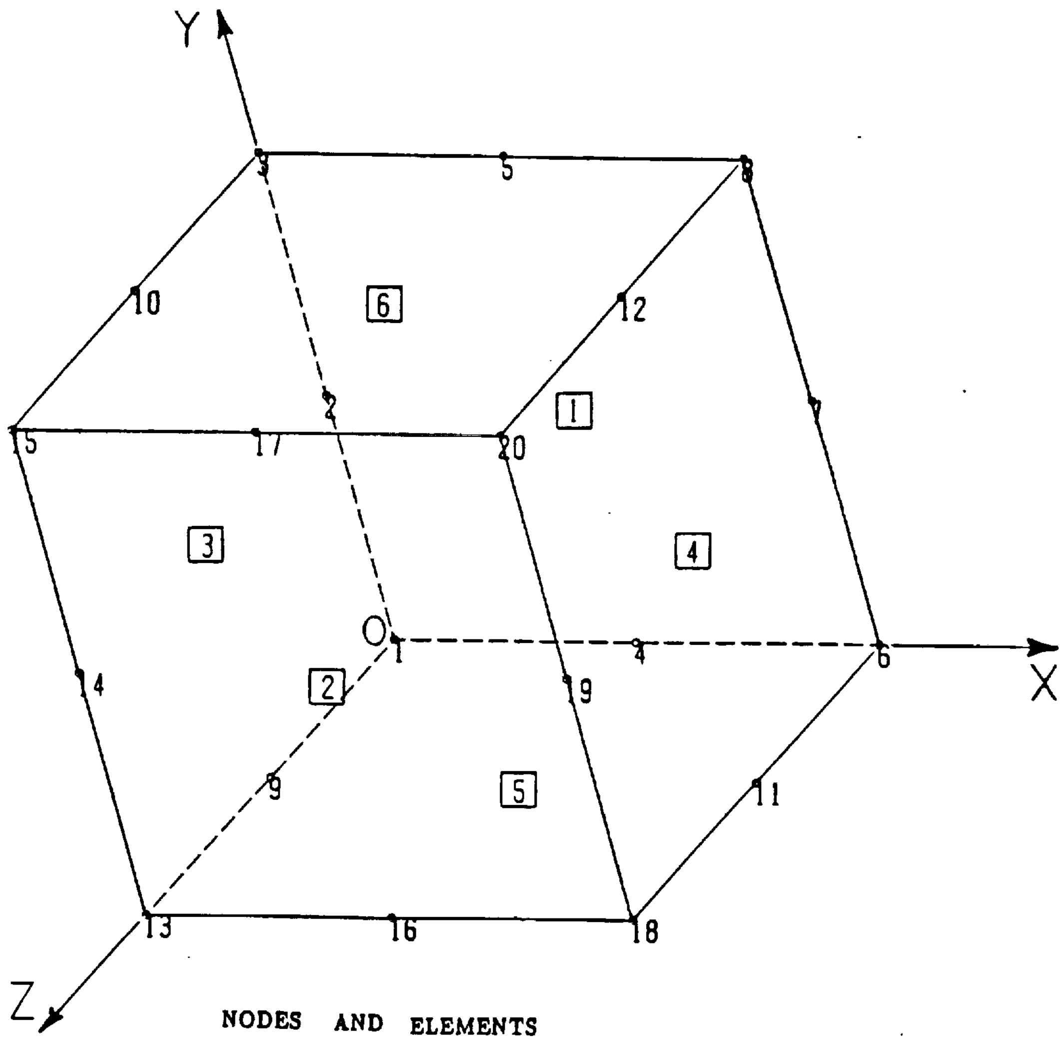
```
1                         BEST NATURAL FREQUENCY ANALYSIS
         JOB TITLE:   FREE VIBRATION OF A 3-D CANTILEVER(1*1*6.5) BY 2 REGION MESH.
                  4 CONVERGED MODE SHAPES IN     9 ITERATIONS
```

MODE NUMBER	FREQUENCY HERTZ	FREQUENCY RAD/SEC
1	0.39055E+00	0.24539E+01
2	0.39055E+00	0.24539E+01
3	0.23424E+01	0.14718E+02
4	0.61734E+01	0.38789E+02

```
         JOB TITLE:   FREE VIBRATION OF A 3-D CANTILEVER(1*1*6.5) BY 2 REGION MESH.
     DEFLECTED SHAPE FOR MODE      1     AT EIGEN FREQUENCY    3.9055E-01 HERTZ        OF REGION = GMR1
```

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	Z-DISPL.	X-TRAC.	Y TRAC.	Z TRAC.
1	1	0.00000E+00	0.00000E+00	0.00000E+00	-0.10331E+01	0.86608E+02	-0.57112E+00
1	8	0.00000E+00	0.00000E+00	0.00000E+00	-0.42060E+01	-0.14985E+02	0.34536E+00
1	7	0.00000E+00	0.00000E+00	0.00000E+00	-0.12496E+01	-0.11635E+03	0.90565E+00
1	6	0.00000E+00	0.00000E+00	0.00000E+00	-0.23565E+01	-0.10224E+03	0.61640E+00
1	5	0.00000E+00	0.00000E+00	0.00000E+00	-0.10331E+01	-0.86608E+02	-0.57125E+00
1	4	0.00000E+00	0.00000E+00	0.00000E+00	-0.42060E+01	0.14965E+02	0.34536E+00
1	3	0.00000E+00	0.00000E+00	0.00000E+00	-0.12496E+01	0.11635E+03	0.90581E+00
1	2	0.00000E+00	0.00000E+00	0.00000E+00	-0.23565E+01	0.10224E+03	0.61643E+00

```
EXAMPLE PROBLEM: HEAT602

ANALYSIS TYPE: STEADY-STATE HEAT CONDUCTION
  3-D ANALYSIS

PROBLEM DESCRIPTION:
  LINEAR VARIATION OF TEMPERATURE IN A CUBE. INTERIOR SAMPLING
  POINTS INCLUDED FOR OUTPUT OF TEMPERATURE AND FLUX.

BOUNDARY ELEMENT MODEL:
  SINGLE GMR, SIX SURFACE ELEMENTS.

REFERENCE FOR ANALYTICAL SOLUTION:
  CARSLAW AND JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP92.

SOLUTION POINTS TO VERIFY:
  GMR      NODE          (TEMPERATURE)
                    ANALYTICAL           BEST
  GMR1      22        25.00             25.00

  GMR      NODE             (FLUX-Y)
                    ANALYTICAL           BEST
  GMR1      22       -2500.0           -2500.2

RUN TIME:
   0.6 X BASE PROBLEM

MISCELLANEOUS:
  INTEGRATION FILES ARE SAVED FOR RESTART.
  BINARY RESULTS FILE IS WRITTEN.
```



NODES AND ELEMENTS

```
**CASE
  TITLE   ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
  HEAT
  RESTART WRITE
  BINARY

**MATE
  ID MAT1
  TEMP  70.0
    COND 25.

**GMR
  ID GMR1
  MAT MAT1
  TREF  70.0
  POINTS
   1       0.0      0.0      0.0
   2       0.0      0.5      0.0
   3       0.0      1.0      0.0
   4       0.5      0.0      0.0
   5       0.5      1.0      0.0
   6       1.0      0.0      0.0
   7       1.0      0.5      0.0
   8       1.0      1.0      0.0
   9       0.0      0.0      0.5
  10       0.0      1.0      0.5
  11       1.0      0.0      0.5
  12       1.0      1.0      0.5
  13       0.0      0.0      1.0
  14       0.0      0.5      1.0
  15       0.0      1.0      1.0
  16       0.5      0.0      1.0
  17       0.5      1.0      1.0
  18       1.0      0.0      1.0
  19       1.0      0.5      1.0
  20       1.0      1.0      1.0
  SURFACE SURF1
    TYPE QUAD
    ELEMENTS
  1     1  2  3  5  8  7  6  4
  2    13 14 15 17 20 19 18 16
  3     1  2  3 10 15 14 13  9
  4     6  7  8 12 20 19 18 11
  5     1  4  6 11 18 16 13  9
  6     3  5  8 12 20 17 15 10
  NORMAL  1 +
  SAMPLING POINTS
       21       0.5    0.0     0.5
       22       0.5    0.25    0.5
       23       0.5    0.5     0.5
       24       0.5    0.75    0.5
       25       0.5    1.0     0.5
```

```
**BCSET
  ID TRAC15
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 5
  TEMP
    SPLIST ALL
    T 1      0.0
**BCSET
  ID TRAC16
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 6
  TEMP
    SPLIST ALL
    T 1    100.0

$   ZERO FLUX BOUNDARY CONDITION ASSUMED ON ELEMENTS  1 THRU 4

$ END OF DATA
```

JOB TITLE: ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
HEAT RATE CALCULATION AT TIME = 1.000000

HEAT RATE FOR REGION GMR1

ELEMENT	HEAT RATE
1	0.00000E+00
2	0.00000E+00
3	0.00000E+00
4	0.00000E+00
5	0.25000E+04
6	-0.25000E+04
TOTAL HEAT RATE	0.00000E+00

JOB TITLE: ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
INTERIOR TEMPERATURE AT TIME = 1.000000 FOR REGION = GMR1

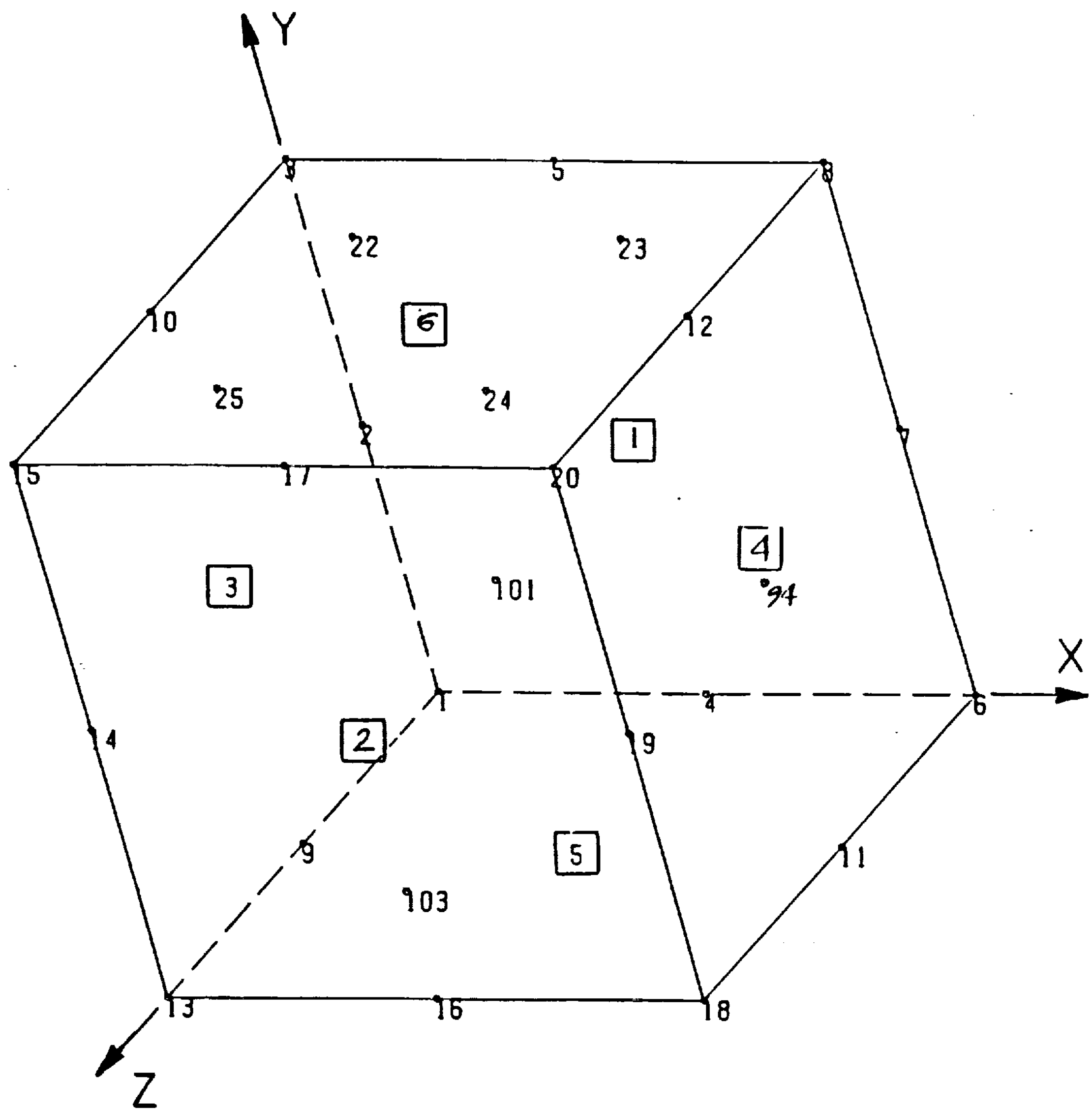
NODE	TEMPERATURE
21	0.000000E+00
22	0.249999E+02
23	0.500009E+02
24	0.750002E+02
25	0.100000E+03

JOB TITLE: ONE-DIMENSIONAL POTENTIAL FLOW IN A CUBE - INTERIOR POINTS
INTERIOR FLUX AT TIME = 1.000000 FOR REGION = GMR1

NODE	FLUX-X	FLUX-Y	FLUX-Z
21	0.000000E+00	-0.249994E+04	0.000000E+00
22	0.142300E-05	-0.250018E+04	0.142300E-05
23	0.150466E-10	-0.250005E+04	0.149631E-10
24	-0.142297E-05	-0.250013E+04	-0.142297E-05
25	0.000000E+00	-0.249994E+04	0.000000E+00

END OF ANALYSIS

```
EXAMPLE PROBLEM: HEAT604

ANALYSIS TYPE: STEADY-STATE HEAT CONDUCTION
  3-D ANALYSIS, MIXTURE OF LINEAR AND QUADRATIC ELEMENTS

PROBLEM DESCRIPTION:
  LINEAR VARIATION OF TEMPERATURE IN A CUBE. INTERIOR SAMPLING
  POINTS INCLUDED FOR OUTPUT OF TEMPERATURE AND FLUX.

BOUNDARY ELEMENT MODEL:
  INCLUDED ARE EIGHT AND NINE NODED SURFACE ELEMENTS, WITH
  BOTH LINEAR AND QUADRATIC FUNCTIONAL VARIATION OF THE
  FIELD VARIABLES.

REFERENCE FOR ANALYTICAL SOLUTION:
  CARSLAW AND JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS, PP92.

SOLUTION POINTS TO VERIFY:
  GMR       NODE          (TEMPERATURE)
                     ANALYTICAL            BEST
  GMR1      101        0.500              0.500

  GMR       NODE              (FLUX-Z)
                     ANALYTICAL            BEST
  GMR1      102       -2.000             -2.000

RUN TIME:
   0.5 X BASE PROBLEM

MISCELLANEOUS:
  SEPARATE SURFACES MUST BE USED FOR THE LINEAR AND QUADRATIC
  ELEMENTS. EVEN THOUGH TWO SURFACES ARE PRESENT, THERE IS ONLY
  ONE 'DISJOINT BOUNDARY'. CONSEQUENTLY, THERE IS ONLY ONE ELEMENT-
  SIGN PAIR ON THE 'NORMAL' INPUT LINE.

  MATERIAL MAT2 IS NOT UTILIZED.
  INTEGRATION FILES ARE SAVED FOR RESTART.
```



NODES AND ELEMENTS

```
**CASE
  TITLE   HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS
$
$    THREE-DIMENSIONAL GEOMETRY (DEFAULT)
$    STEADY-STATE HEAT TRANSFER
  HEAT
$    SAVE INTEGRATION FT FILES FOR LATER USE
  RESTART WRITE
$    PRINT INTERIOR TEMP AND FLUX


**MATE
  ID MAT1
$    REFERENCE TEMPERATURE
  TEMP   0.0
$    THERMAL CONDUCTIVITY
    COND  1.


**MATE
  ID MAT2
$    REFERENCE TEMPERATURE
  TEMP   0.0
$    THERMAL CONDUCTIVITY
    COND  1.


**GMR
  ID GMR1
$    USE MATERIAL MAT1 FOR REGION
  MAT MAT1
$    REFERENCE TEMPERATURE FOR REGION
  TREF   0.0
$
  POINTS
$ ID         X         Y         Z
   1       0.0       0.0       0.0
   2       0.0    .25000       0.0
   3       0.0    .50000       0.0
   4    .25000       0.0       0.0
   5    .25000    .50000       0.0
   6    .50000       0.0       0.0
   7    .50000    .25000       0.0
   8    .50000    .50000       0.0
   9       0.0       0.0    .25000
  10       0.0    .50000    .25000
  11    .50000       0.0    .25000
  12    .50000    .50000    .25000
  13       0.0       0.0    .50000
  14       0.0    .25000    .50000
  15       0.0    .50000    .50000
  16    .25000       0.0    .50000
  17    .25000    .50000    .50000
  18    .50000       0.0    .50000
```

```
   19     .50000    .25000    .50000
   20     .50000    .50000    .50000
   21     .25000    .50000    .25000
   22     .12500    .50000    .12500
   23     .37500    .50000    .12500
   24     .37500    .50000    .37500
   25     .12500    .50000    .37500
   92     .25000    .25000    .50000
   94     .50000    .25000    .25000
$
  SURFACE SURF1
$    USE LINEAR FUNCTIONAL VARIATION
    TYPE LINEAR
$
    ELEMENTS
$  ID    G1 G2 G3 G4 G5 G6 G7 G8 G9
    2    13 14 15 17 20 19 18 16 92
$
  SURFACE SURF2
$
$    USE QUADRATIC FUNCTIONAL VARIATION
    TYPE QUADRATIC
$
    ELEMENTS
$  ID    G1 G2 G3 G4 G5 G6 G7 G8 G9
    1     1  2  3  5  8  7  6  4
    3     1  2  3 10 15 14 13  9
    4     6  7  8 12 20 19 18 11 94
    5     1  4  6 11 18 16 13  9
    6     3  5  8 12 20 17 15 10

$    OUTWARD (RIGHT-HAND) NORMAL OF ELEMENT 1 IS POSITIVE
$     (ONLY ONE DISJOINT BOUNDARY)
  NORMAL  1 +


  SAMPLING  POINTS
     101     0.25    0.25    0.25
     102     0.25    0.25    0.50
     103     0.25    0.10    0.50
$
$
$     SPECIFY TEMPERATURE = 0.0 AT Z=0.0 (ELEMENT 1, GMR1)
$
**BCSET
  ID TEMP11
  VALUE
  GMR GMR1
  SURFACE SURF2
    ELEMENTS 1
  TEMP
    SPLIST ALL
    T 1     0.0
$
$     SPECIFY TEMPERATURE = 1.0 AT Z=0.5 (ELEMENT 2, GMR1)
$
**BCSET
  ID TEMP12
```

```
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 2
  TEMP
    SPLIST ALL
    T 1      1.0
$
$     REMAINING ELEMENTS HAVE ZERO FLUX BOUNDARY CONDITIONS (DEFAULT)
$
$
$  END OF DATA
```

JOB TITLE: HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS
INTERIOR TEMPERATURE AT TIME = 1.000000 FOR REGION = GMR1

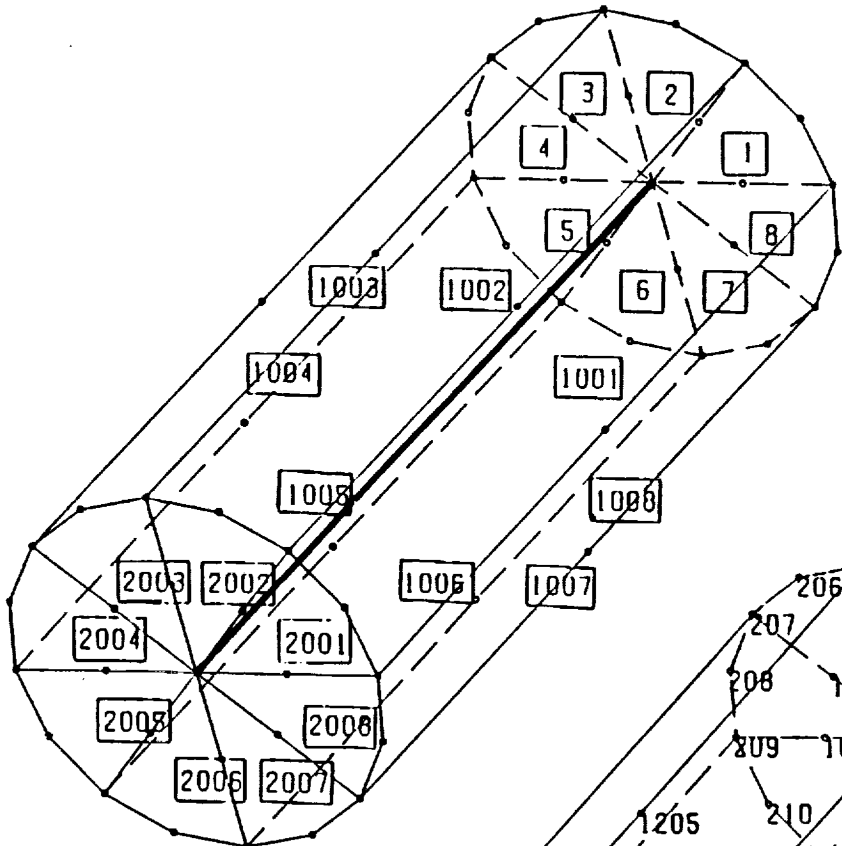
NODE	TEMPERATURE
101	0.500011E+00
102	0.100000E+01
103	0.100000E+01

JOB TITLE: HEAT TRANSFER WITH ADJACENT LINEAR-QUADRATIC ELEMENTS
INTERIOR FLUX AT TIME = 1.000000 FOR REGION = GMR1

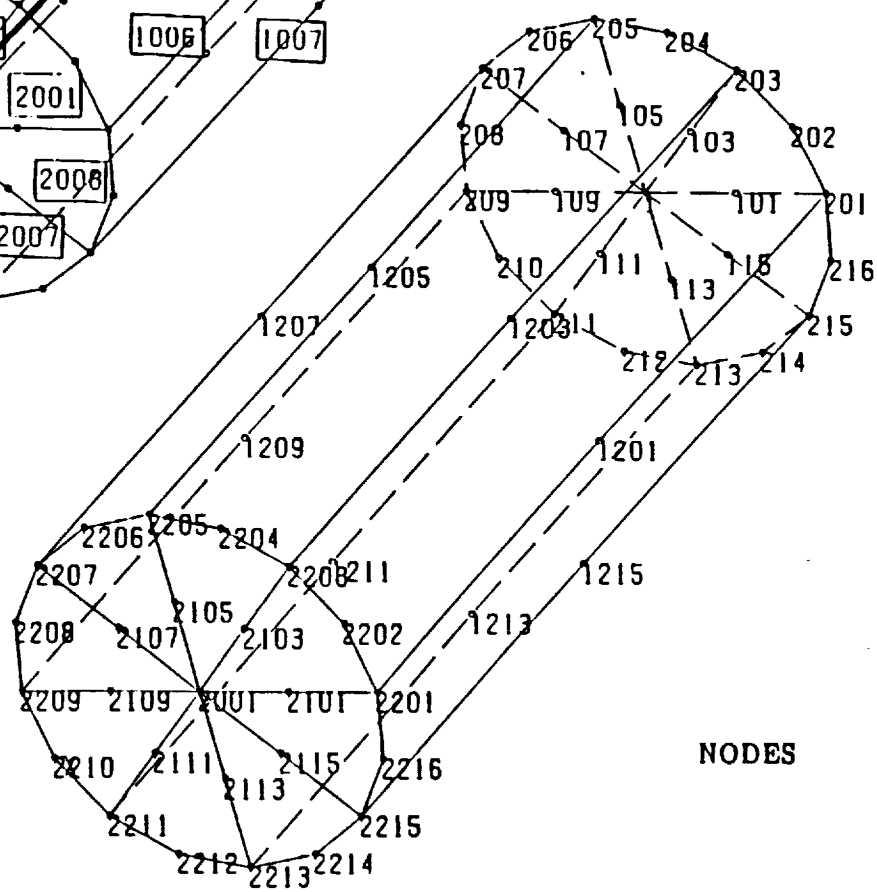
NODE	FLUX-X	FLUX-Y	FLUX-Z
101	0.670418E-07	0.187336E-08	-0.200004E+01
102	0.000000E+00	0.000000E+00	-0.200004E+01
103	0.000000E+00	0.000000E+00	-0.200004E+01

END OF ANALYSIS

```
EXAMPLE PROBLEM: HEAT605

ANALYSIS TYPE: TRANSIENT HEAT CONDUCTION
  3-D ANALYSIS

PROBLEM DESCRIPTION:
  TRANSIENT HEAT FLOW BETWEEN TWO CONCENTRIC CIRCULAR CYLINDERS.
  AT TIME ZERO, THE TEMPERATURE OF THE INNER CYLINDER IS ELEVATED
  FROM ZERO TO 1.0. THE RESPONSE IS MONITORED AT A NUMBER OF
  INTERIOR SAMPLING POINTS.

BOUNDARY ELEMENT MODEL:
  THE OUTER CYLINDER HAS RADIUS 8.0 AND IS MODELED WITH 3-D
  SURFACE ELEMENTS. THE INNER CYLINDER (RADIUS=1.0) IS REPRESENTED
  BY HOLE ELEMENTS.

REFERENCE FOR ANALYTICAL SOLUTION:
  CARSLAW AND JAEGER (1959), CONDUCTION OF HEAT IN SOLIDS,
    PP205-207.

SOLUTION POINTS TO VERIFY:
  GMR      NODE    TIME                 (TEMPERATURE)
                              ANALYTICAL              BEST
  GMR1     5005    4.00         0.1105              0.1067
                   8.00         0.1971              0.1948

RUN TIME:
   29 X BASE PROBLEM

MISCELLANEOUS:
  THE CYLINDERS HAVE A LENGTH OF 40.0 UNITS. CONSEQUENTLY, NEARLY
  PLANAR CONDITIONS EXIST AT Z=20.0, WHERE THE SAMPLING POINTS
  ARE LOCATED.
  TOTAL HEAT RATE CALCULATION DOES NOT INCLUDE CONTRIBUTION
  FROM HOLE ELEMENTS.
  A BINARY OUTPUT FILE IS SAVED.
```



ELEMENTS



NODES

```
**CASE
  TITLE   TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT
$
$    THREE-DIMENSIONAL GEOMETRY (DEFAULT)
$    TRANSIENT HEAT TRANSFER
  HEAT TRANSIENT     8    1.0
  BINARY


**MATE
  ID MAT1
$    REFERENCE TEMPERATURE
  TEMP   0.0
$    THERMAL CONDUCTIVITY
    COND 1.0
$    DENSITY
    DENS 1.0
$    SPECIFIC HEAT
    SPEC 1.0


**GMR
  ID GMR1
$    USE MATERIAL MAT1 FOR REGION
  MAT MAT1
$    REFERENCE TEMPERATURE FOR REGION
  TREF   0.0
$
  POINTS
$       ID           X            Y        Z
$
         1        0.00000      0.00000    0.0
       101        4.00000      0.00000    0.0
       103        2.82843      2.82843    0.0
       105        0.00000      4.00000    0.0
       107       -2.82843      2.82843    0.0
       109       -4.00000      0.00000    0.0
       111       -2.82843     -2.82843    0.0
       113        0.00000     -4.00000    0.0
       115        2.82843     -2.82843    0.0
       201        8.00000      0.00000    0.0
       202        7.39104      3.06147    0.0
       203        5.65685      5.65685    0.0
       204        3.06147      7.39104    0.0
       205        0.00000      8.00000    0.0
       206       -3.06147      7.39104    0.0
       207       -5.65685      5.65685    0.0
       208       -7.39104      3.06147    0.0
       209       -8.00000      0.00000    0.0
       210       -7.39104     -3.06147    0.0
       211       -5.65685     -5.65685    0.0
       212       -3.06147     -7.39104    0.0
       213        0.00000     -8.00000    0.0
       214        3.06147     -7.39104    0.0
```

```
        215         5.65685      -5.65685    0.0
        216         7.39104      -3.06147    0.0
$
       1201         8.00000       0.00000   20.0
       1203         5.65685       5.65685   20.0
       1205         0.00000       8.00000   20.0
       1207        -5.65685       5.65685   20.0
       1209        -8.00000       0.00000   20.0
       1211        -5.65685      -5.65685   20.0
       1213         0.00000      -8.00000   20.0
       1215         5.65685      -5.65685   20.0
$
       2001         0.00000       0.00000   40.0
       2101         4.00000       0.00000   40.0
       2103         2.82843       2.82843   40.0
       2105         0.00000       4.00000   40.0
       2107        -2.82843       2.82843   40.0
       2109        -4.00000       0.00000   40.0
       2111        -2.82843      -2.82843   40.0
       2113         0.00000      -4.00000   40.0
       2115         2.82843      -2.82843   40.0
       2201         8.00000       0.00000   40.0
       2202         7.39104       3.06147   40.0
       2203         5.65685       5.65685   40.0
       2204         3.06147       7.39104   40.0
       2205         0.00000       8.00000   40.0
       2206        -3.06147       7.39104   40.0
       2207        -5.65685       5.65685   40.0
       2208        -7.39104       3.06147   40.0
       2209        -8.00000       0.00000   40.0
       2210        -7.39104      -3.06147   40.0
       2211        -5.65685      -5.65685   40.0
       2212        -3.06147      -7.39104   40.0
       2213         0.00000      -8.00000   40.0
       2214         3.06147      -7.39104   40.0
       2215         5.65685      -5.65685   40.0
       2216         7.39104      -3.06147   40.0
$
$
  SURFACE SURF1
$    USE QUADRATIC FUNCTIONAL VARIATION
    TYPE QUAD
$
$    DISC AT NEAR END OF CYLINDER
$
    ELEMENTS
$ID        G1   G2   G3   G4   G5   G6    G7    G8
   1        1  101  201  202  203  103
   2        1  103  203  204  205  105
   3        1  105  205  206  207  107
   4        1  107  207  208  209  109
   5        1  109  209  210  211  111
   6        1  111  211  212  213  113
   7        1  113  213  214  215  115
   8        1  115  215  216  201  101
$
$    SURFACE OF OUTER CYLINDER
$
```

```
1001      201  202  203 1203 2203 2202 2201 1201
1002      203  204  205 1205 2205 2204 2203 1203
1003      205  206  207 1207 2207 2206 2205 1205
1004      207  208  209 1209 2209 2208 2207 1207
1005      209  210  211 1211 2211 2210 2209 1209
1006      211  212  213 1213 2213 2212 2211 1211
1007      213  214  215 1215 2215 2214 2213 1213
1008      215  216  201 1201 2201 2216 2215 1215
$
$    DISC AT FAR END OF CYLINDER
$
2001     2001 2101 2201 2202 2203 2103
2002     2001 2103 2203 2204 2205 2105
2003     2001 2105 2205 2206 2207 2107
2004     2001 2107 2207 2208 2209 2109
2005     2001 2109 2209 2210 2211 2111
2006     2001 2111 2211 2212 2213 2113
2007     2001 2113 2213 2214 2215 2115
2008     2001 2115 2215 2216 2201 2101
$
$    OUTWARD (RIGHT-HAND) NORMAL OF ELEMENT    1 IS NEGATIVE
  NORMAL      1 -
$
$    SURFACE OF INNER CYLINDER
$
 HOLE
   POINTS
9001         0.0        0.0       5.0
9002         0.0        0.0       7.5
9003         0.0        0.0      10.0
9004         0.0        0.0      15.0
9005         0.0        0.0      20.0
9006         0.0        0.0      25.0
9007         0.0        0.0      30.0
9008         0.0        0.0      32.5
9009         0.0        0.0      35.0
$
$    USE QUADRATIC FUNCTIONAL VARIATION FOR 3-NODED HOLE ELEMENTS
    TYPE QUAD
$
    ELEMENTS
$       ID     RAD     G1   G2   G3
      9001    1.00   9001 9002 9003
      9002    1.00   9003 9004 9005
      9003    1.00   9005 9006 9007
      9004    1.00   9007 9008 9009
$

  SAMPLING  POINTS
5001         1.7        0.0       20.0
5002         2.4        0.0       20.0
5003         3.1        0.0       20.0
5004         3.8        0.0       20.0
5005         4.5        0.0       20.0
5006         5.2        0.0       20.0
5007         5.9        0.0       20.0
5008         6.6        0.0       20.0
5009         7.3        0.0       20.0
```

```
$
$     SPECIFY TEMPERATURE = 1.0 ON INNER CYLINDER
$
**BCSET
  ID TEMP1
$       VALUE BOUNDARY CONDITION SET
  VALUE
  GMR GMR1
  HOLE
    ELEMENTS  9001  9002  9003  9004
  TEMP
    SPLIST ALL
    T 1     1.0
$
$     SPECIFY TEMPERATURE = 0.0 ON OUTER CYLINDER
$
**BCSET
  ID TEMP2
$       VALUE BOUNDARY CONDITION SET
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS  1001  1002  1003  1004
    ELEMENTS  1005  1006  1007  1008
  TEMP
    SPLIST ALL
    T 1     0.0
$
$
$     ENDS HAVE ZERO FLUX BOUNDARY CONDITIONS (DEFAULT)

$ END OF DATA
```

```
JOB TITLE:    TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT
HOLE ELEMENT SOLUTION AT TIME =      8.000000          FOR REGION = GMR1
```

ELEMENT	NODE NO.	TEMPERATURE	FLUX
9001	9001	0.10000E+01	-0.85685E+00
9001	9002	0.10000E+01	-0.64049E+00
9001	9003	0.10000E+01	-0.58113E+00
9002	9003	0.10000E+01	-0.58113E+00
9002	9004	0.10000E+01	-0.57257E+00
9002	9005	0.10000E+01	-0.57366E+00
9003	9005	0.10000E+01	-0.57366E+00
9003	9006	0.10000E+01	-0.57257E+00
9003	9007	0.10000E+01	-0.58113E+00
9004	9007	0.10000E+01	-0.58113E+00
9004	9008	0.10000E+01	-0.64049E+00
9004	9009	0.10000E+01	-0.85685E+00

```
JOB TITLE:    TRANSIENT FLOW BETWEEN TWO CYLINDERS - HOLE ELEMENT
INTERIOR TEMPERATURE AT TIME =      8.000000          FOR REGION = GMR1
```

NODE	TEMPERATURE
5001	0.700817E+00
5002	0.509723E+00
5003	0.373628E+00
5004	0.272088E+00
5005	0.194836E+00
5006	0.135517E+00
5007	0.896075E-01
5008	0.535336E-01
5009	0.000000E+00

END OF ANALYSIS

EXAMPLE PROBLEM: PLAS602

ANALYSIS TYPE: STRESS ANALYSIS
3-D, ELASTOPLASTIC ANALYSIS, VON MISES MATERIAL MODEL WITH PERFECTLY PLASTIC BEHAVIOR, VARIABLE STIFFNESS ALGORITHM.

PROBLEM DESCRIPTION :
THICK CYLINDER SUBJECTED TO INCREASING INTERNAL PRESSURE, UNDER PLANE STRAIN CONDITIONS.

BOUNDARY ELEMENT MODEL:
15-DEGREE SEGMENT, WITH 1:2 RATIO, 1-GMR, 18 BOUNDARY ELEMENTS, 4 (TWENTY-NODED) ISOPARAMETRIC CELLS.

REFERENCE FOR ANALYTICAL SOLUTION:
HILL,R.(1950) THE MATHEMATICAL THEORY OF PLASTICITY,PG.106-111 PROVIDES ANALYTICAL SOLUTION TO THIS PROBLEM.

SOLUTION POINTS TO VERIFY:

RADIAL DISPLACEMENT (U-X) AT THE OUTER SURFACE

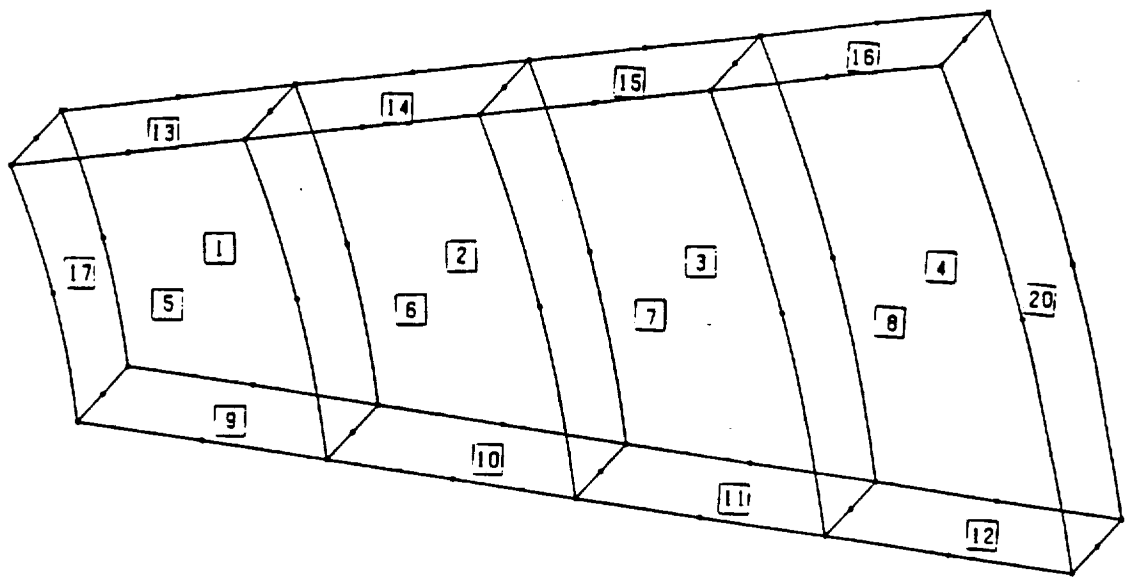
TIME	NODE	BEST SOLN.	ANALYTICAL SOLN.
2.0	10	1.2925	1.2921
4.0	10	1.5905	1.5855
10.0	10	2.7196	2.7279

HOOP STRESS (SIG-YY) ALONG THE THICKNESS OF THE CYLINDER

TIME	NODE	BEST SOLN.	ANALYTICAL SOLN.
2.0	20	428.1	433.0
2.0	10	184.6	183.2
2.0	22	257.5	250.6
10.0	20	284.7	280.4
10.0	10	388.9	389.7
10.0	22	542.2	541.3

RUN TIME:
31 X BASE PROBLEM

MISCELLENEOUS :
LOAD-DISPLACEMENT RESPONSE (NORMALISED RADIAL DISPLACEMENT AS) (A FUNCTION OF INTERNAL PRESSURE) AND DISTRIBUTION OF CIRCUMFERENTIAL STRESS THROUGH THE THICKNESS OF THE CYLINDER ARE THE PRIMARY INTERESTS OF THIS ANALYSIS.



ELEMENTS

```
**CASE
  TITLE  THICK CYLINDER UNDER INTERNAL PRESSURE
  PLASTICITY  DIRECT        $ SPECIFIES VARIABLE STIFFNESS PLASTICITY
  TIMES   1.  2.  3.  4.  5.  6.  7.  8.  9.  10.

**MATE
  ID MAT1
    TEMP 70.0
    EMOD 2600.0
    POIS 0.3
  INELASTIC
    VON MISES
    YIELD 600.0
    CURVE
      600.0 10.0
$ (PERFECTLY PLASTIC MATERIAL WITH ZERO STRAIN HARDENING)

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  POINTS
    1      9.9144    -1.3053    0.0
    2     11.1538    -1.4684    0.0
    3     12.3931    -1.6316    0.0
    4     13.6324    -1.7947    0.0
    5     14.8717    -1.9579    0.0
    6     16.1110    -2.1211    0.0
    7     17.3503    -2.2842    0.0
    8     18.5896    -2.4474    0.0
    9     19.8289    -2.6105    0.0
   10     20.0        0.0       0.0
   11     19.8289     2.6105    0.0
   12     18.5896     2.4474    0.0
   13     17.3503     2.2842    0.0
   14     16.1110     2.1211    0.0
   15     14.8717     1.9579    0.0
   16     13.6324     1.7947    0.0
   17     12.3931     1.6316    0.0
   18     11.1538     1.4684    0.0
   19      9.9144     1.3053    0.0
   20     10.0        0.0       0.0
   21     12.5        0.0       0.0
   22     15.0        0.0       0.0
   23     17.5        0.0       0.0
  101      9.9144    -1.3053    1.0
  102     11.1538    -1.4684    1.0
  103     12.3931    -1.6316    1.0
  104     13.6324    -1.7947    1.0
  105     14.8717    -1.9579    1.0
  106     16.1110    -2.1211    1.0
  107     17.3503    -2.2842    1.0
```

```
    108        18.5896      -2.4474      1.0
    109        19.8289      -2.6105      1.0
    110        20.0          0.0         1.0
    111        19.8289       2.6105      1.0
    112        18.5896       2.4474      1.0
    113        17.3503       2.2842      1.0
    114        16.1110       2.1211      1.0
    115        14.8717       1.9579      1.0
    116        13.6324       1.7947      1.0
    117        12.3931       1.6316      1.0
    118        11.1538       1.4684      1.0
    119         9.9144       1.3053      1.0
    120        10.0          0.0         1.0
    121        12.5          0.0         1.0
    122        15.0          0.0         1.0
    123        17.5          0.0         1.0
    201         9.9144      -1.3053      0.5
    203        12.3931      -1.6316      0.5
    205        14.8717      -1.9579      0.5
    207        17.3503      -2.2842      0.5
    209        19.8289      -2.6105      0.5
    211        19.8289       2.6105      0.5
    213        17.3503       2.2842      0.5
    215        14.8717       1.9579      0.5
    217        12.3931       1.6316      0.5
    219         9.9144       1.3053      0.5
  SURFACE SURF1
     TYPE QUAD
     ELEMENTS
   1      1    20    19    18    17    21     3     2
   2      3    21    17    16    15    22     5     4
   3      5    22    15    14    13    23     7     6
   4      7    23    13    12    11    10     9     8
   5    101   102   103   121   117   118   119   120
   6    103   104   105   122   115   116   117   121
   7    105   106   107   123   113   114   115   122
   8    107   108   109   110   111   112   113   123
   9    101   201     1     2     3   203   103   102
  10    103   203     3     4     5   205   105   104
  11    105   205     5     6     7   207   107   106
  12    107   207     7     8     9   209   109   108
  13     19   219   119   118   117   217    17    18
  14     17   217   117   116   115   215    15    16
  15     15   215   115   114   113   213    13    14
  16     13   213   113   112   111   211    11    12
  17      1   201   101   120   119   219    19    20
  20    109   209     9    10    11   211   111   110
  NORMAL 1 +
  VOLUME
    TYPE QUAD
    CELL
 51 101 201 1 20 19 219 119 120 102 2 18 118 103 203 3 21 17 217 117 121
 52 103 203 3 21 17 217 117 121 104 4 16 116 105 205 5 22 15 215 115 122
 53 105 205 5 22 15 215 115 122 106 6 14 114 107 207 7 23 13 213 113 123
 54 107 207 7 23 13 213 113 123 108 8 12 112 109 209 9 10 11 211 111 110
FULL
```

```
**BCSET
  ID DISP13
  VALUE
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 1 2 3 4 5 6 7 8
    DISP 3
    SPLIST ALL
       T 1 0.0

**BCSET
  ID DISP73
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 9 10 11 12 13 14 15 16
    DISP 1
    SPLIST ALL
       T 1 0.0

**BCSET
  ID TRAC12
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 17
    TIMES 1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12.
    TIMES 13. 14. 15. 16. 17. 18. 19. 20.
    TRAC 1
    SPLIST ALL
      T  1 -250.0
      T  2 -275.0
      T  3 -300.0
      T  4 -325.0
      T  5 -350.0
      T  6 -375.0
      T  7 -400.0
      T  8 -410.0
      T  9 -420.0
      T 10 -430.0
      T 11 -440.0
      T 12 -450.0
      T 13 -455.0
      T 14 -460.0
      T 15 -465.0
      T 16 -470.0
      T 17 -475.0
      T 18 -480.0
      T 19 -485.0
      T 20 -495.0
$
$  END OF DATA
```

JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE
INTERIOR STRESS AT TIME = 10.0000 FOR REGION = GMR1

NODE	SIGMA-XX	SIGMA-YY	SIGMA-ZZ	TAU-XY	TAU-XZ	TAU-YZ
101	-0.397758E+03	0.270349E+03	-0.437602E+02	0.909978E+02	-0.231789E+01	0.305473E+00
201	-0.384075E+03	0.285010E+03	-0.330210E+02	0.894291E+02	0.105515E+01	-0.191894E+00
1	-0.389428E+03	0.278187E+03	-0.363330E+02	0.918805E+02	0.412454E+01	-0.543628E+00
20	-0.407711E+03	0.284736E+03	-0.409720E+02	0.414813E+00	0.000000E+00	-0.127245E-01
19	-0.395556E+03	0.272460E+03	-0.406881E+02	-0.910941E+02	0.243674E+01	0.321175E+00
219	-0.385773E+03	0.283308E+03	-0.344642E+02	-0.894289E+02	0.159989E+00	-0.314562E-01
119	-0.393616E+03	0.274565E+03	-0.387867E+02	-0.908113E+02	-0.217849E+01	-0.287120E+00
120	-0.411829E+03	0.280581E+03	-0.441993E+02	0.121263E+00	0.000000E+00	0.132679E+00

JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE
INTERIOR STRESS AT TIME = 10.0000 FOR REGION = GMR1

NODE	SIGMA-XX	SIGMA-YY	SIGMA-ZZ	TAU-XY	TAU-XZ	TAU-YZ
109	0.114888E+02	0.387736E+03	0.129299E+03	0.504143E+02	0.408951E+00	-0.539817E-01
209	0.510522E+01	0.377527E+03	0.114790E+03	0.497664E+02	0.102944E+00	-0.131649E-01
9	0.197399E+02	0.409735E+03	0.123835E+03	0.523260E+02	0.254109E-01	-0.341142E-02
10	0.291534E+01	0.388948E+03	0.115917E+03	0.586847E-02	0.000000E+00	-0.674978E-02
11	0.205498E+02	0.411276E+03	0.124728E+03	-0.524186E+02	-0.231390E-01	-0.299907E-02
211	0.527742E+01	0.378152E+03	0.115029E+03	-0.498270E+02	0.698976E-01	0.987533E-02
111	0.115394E+02	0.388292E+03	0.129467E+03	-0.504840E+02	0.423199E+00	0.558628E-01
110	0.192720E+01	0.391854E+03	0.126318E+03	-0.110083E-01	0.000000E+00	-0.552908E-03

JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE
INELASTIC SOLUTION AT TIME = 10.0000

NODE TYPE: E = ELASTIC NODE , P = PLASTIC NODE

NODE	TYPE	EQUIVALENT PLASTIC STRAIN	EQUIVALENT STRESS
101	P	3.38405E-01	6.00026E+02
201	P	3.41655E-01	6.00021E+02
1	P	3.33060E-01	6.00027E+02
20	P	3.41273E-01	6.00028E+02
19	P	3.39528E-01	6.00027E+02
219	P	3.45694E-01	6.00021E+02
119	P	3.41131E-01	6.00028E+02
120	P	3.42517E-01	6.00028E+02

JOB TITLE: THICK CYLINDER UNDER INTERNAL PRESSURE
INELASTIC SOLUTION AT TIME = 10.0000

NODE TYPE: E = ELASTIC NODE , P = PLASTIC NODE

NODE	TYPE	EQUIVALENT PLASTIC STRAIN	EQUIVALENT STRESS
8	E	0.00000E+00	3.99694E+02
12	E	0.00000E+00	3.98760E+02
112	E	0.00000E+00	3.94061E+02
109	E	0.00000E+00	3.44588E+02
209	E	0.00000E+00	3.42505E+02
9	E	0.00000E+00	3.61315E+02
10	E	0.00000E+00	3.43756E+02
11	E	0.00000E+00	3.62019E+02
211	E	0.00000E+00	3.42937E+02
111	E	0.00000E+00	3.45060E+02
110	E	0.00000E+00	3.44982E+02

EXAMPLE PROBLEM: PLAS603

ANALYSIS TYPE: STRESS ANALYSIS
3-D, ELASTOPLASTIC ANALYSIS, TWO-SURFACE CYCLIC PLASTICITY MATERIAL MODEL, ITERATIVE ALGORITHM.

PROBLEM DESCRIPTION:
UNIT CUBE SUBJECTED TO CYCLIC LOADING, SIMPLY SUPPORTED BOUNDARY CONDITIONS, AXIAL TENSION UP TO 1.75 TIMES YIELD STRESS AND AXIAL COMPRESSION UP TO 1.25 TIMES YIELD STRESS APPLIED FOR ONE COMPLETE CYCLE.

BOUNDARY ELEMENT MODEL:
6 (8-NODED) ELEMENTS AROUND BOUNDARY, 1 ON EACH SIDE OF THE UNIT CUBE, 1 (20-NODED) VOLUME CELL FILLING UP THE COMPLETE VOLUME OF THE BODY, GLOBAL BOUNDARY CONDITIONS SPECIFIED EVERYWHERE.

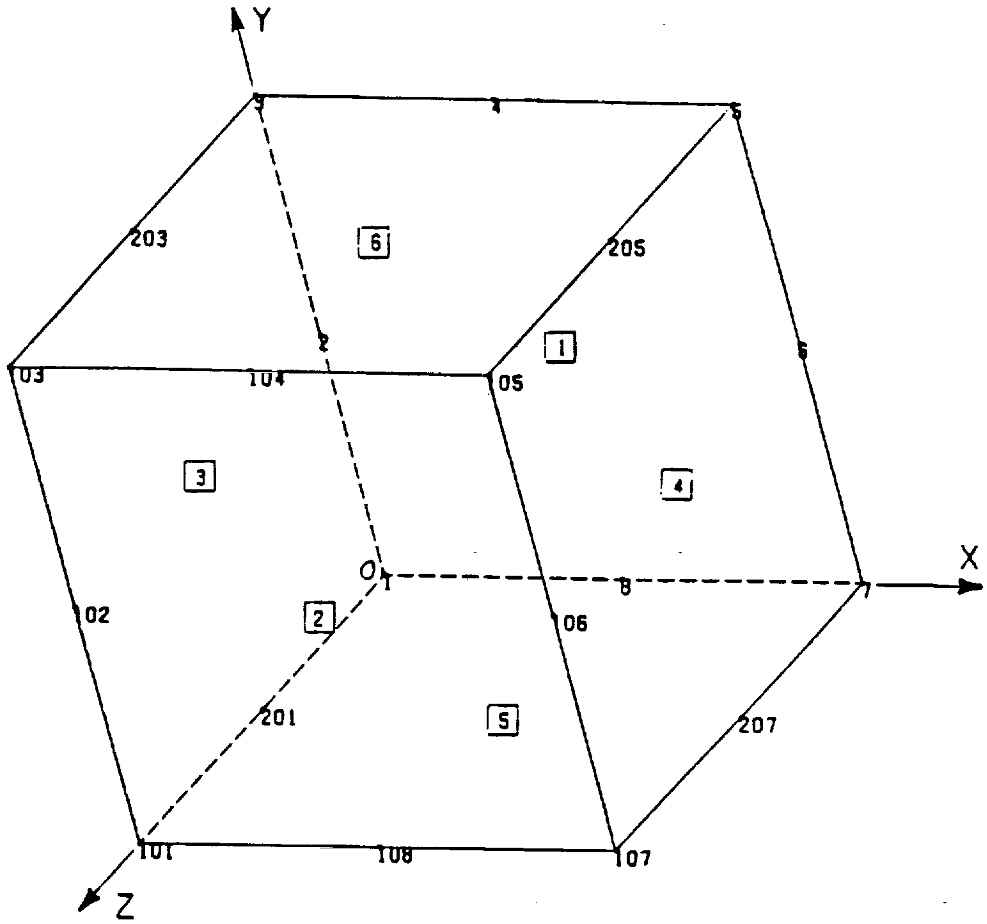
REFERENCE FOR EXPERIMENTAL SOLUTION:
DOMAS,SHARPE,WARD AND YAU(1982), NASA CR-165571.
PRESENT EXPERIMENTAL RESULTS FOR TESTS CONDUCTED PRIMARILY ON A BENCHMARK NOTCH SPECIMEN BUT ALSO PRESENT RESULTS COMPARED HERE FOR A SPECIMEN WITHOUT A NOTCH.

SOLUTION POINTS TO VERIFY:

		STRAIN (EPS-ZZ)	
TIME	NODE	BEST SOLN.	EXPERIMENTAL
10.0	1	0.100268E-01	0.11200E-01
30.0	1	-0.588880E-02	-0.62200E-02

RUN TIME:
6 X BASE PROBLEM

MISCELLANOEUS:
TWO-SURFACE PLASTICITY MODEL DICTATES THAT THE MATERIAL BECOMES PLASTIC AT LOWER STRESS LEVEL ALLOWING FOR A SMOOTH TRANSITION FROM AN ELASTIC TO PLASTIC STATE, IT ALSO HELPS IN DESCRIBING CYCLIC BEHAVIOR BY TAKING INTO ACCOUNT THE REDUCED YIELD STRESS DURING UNLOADING AS SHOWN BY THE BAUSCHINGER EFFECT, THE HYSTERESIS LOOP OF STRESS VS. STRAIN IS OF MAIN INTEREST IN THE RESULTS.



NODES AND ELEMENTS

```
**CASE
  TITLE   CYCLIC LOADING OF UNIT CUBE
  PLASTICITY ITERATIVE      $ ITERATIVE PLASTICITY ALGORITHM SELECTED
  TIMES   4.0 6.0 7.0 8.0 9.0 10.0 12.0 14.0 16.0 18.0 20.0 22.0
  TIMES   24.0 26.0 28.0 30.0 33.0 36.0 38.0 40.0

**MATE
  ID MAT1
$ (ALL UNITS ARE IN KSI)
  TEMP   70.0
    EMOD 23.8E+03
    POIS 0.3
  INELASTIC
    TWO SURFACE              $ TWO-SURFACE MATERIAL MODEL SELECTED
    YIELD 100.0   400.0      $ SPECIFIES INNER AND OUTER YIELD
                             $  STRESS VALUES, RESPECTIVELY
    HARD  35.7E+03   11.9E+03   $ SPECIFIES INNER AND OUTER
                                $  HARDENING VALUES, RESPECTIVELY
   TIME 7 17                 $ SPECIFIES THE TIME STEP NUMBERS AT
                             $  WHICH A REVERSAL OF LOADING TAKES PLACE,
                             $  THIS TURNS OFF THE ACCELERATED ALGORITHM

**GMR
  ID GMR1
  MAT MAT1
  TREF   70.0
  POINTS
   1        0.0      0.0      0.0
   2        0.0      0.5      0.0
   3        0.0      1.0      0.0
   4        0.5      1.0      0.0
   5        1.0      1.0      0.0
   6        1.0      0.5      0.0
   7        1.0      0.0      0.0
   8        0.5      0.0      0.0
 101        0.0      0.0      1.0
 102        0.0      0.5      1.0
 103        0.0      1.0      1.0
 104        0.5      1.0      1.0
 105        1.0      1.0      1.0
 106        1.0      0.5      1.0
 107        1.0      0.0      1.0
 108        0.5      0.0      1.0
 201        0.0      0.0      0.5
 203        0.0      1.0      0.5
 205        1.0      1.0      0.5
 207        1.0      0.0      0.5
  SURFACE SURF1
    TYPE QUAD
    ELEMENTS
 1     1   2   3   4   5   6   7   8
 2    101 108 107 106 105 104 103 102
```

```
3    1  201 101 102 103 203 3 2
4    7  6  5  205 105 106 107 207
5    1  8  7  207 107 108 101 201
6    3  203 103 104 105 205 5  4
NORMAL  1 +
VOLUME
 CELL
51  1 2 3 4 5 6 7 8 201 203 205 207 101 102 103 104 105 106 107 108
FULL
$ (FULL DEFINES THAT THE VOLUME IS COMPLETELY FILLED WITH CELLS)
$ (AND INITIAL STRESS EXPANSION IS USED TO COMPUTE JUMP TERMS.)


**BCSET
  ID DISP13
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 1
  DISP 3
    SPLIST ALL
    T 1 0.0

**BCSET
  ID DISP31
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 3
  DISP 1
    SPLIST ALL
    T 1 0.0

**BCSET
  ID DISP52
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 5
  DISP 2
    SPLIST ALL
    T 1 0.0

**BCSET
  ID TRAC12
  VALUE
  GMR GMR1
  SURFACE SURF1
    ELEMENTS 2
    TIMES 0.0 10.0 30.0 40.0
  TRAC 3
    SPLIST ALL
    T 1     0.0
    T 2   175.0
    T 3  -125.0
    T 4     0.0
$ (TRACTION BOUNDARY COND. SPECIFYING INITIAL LOADING UPTO)
$ (TIME 10.0 TO STRESS OF 175.0 KSI, FOLLOWED BY UNLOADING)
```

```
$ (TO -125.0 KSI AT TIME 30.0 AND FINALLY RELOADED TO 0.0)
$ (AT TIME 40.0.)
$
$  END OF DATA
```

```
JOB TITLE:    CYCLIC LOADING OF UNIT CUBE

NUMBER OF SUB-INCREMENTS =   7  FOR TIME =    30.0000

  NUMBER OF ITERATIONS =   0    FOR SUB-INCREMENT   1
  NUMBER OF ITERATIONS =   1    FOR SUB-INCREMENT   2
  NUMBER OF ITERATIONS =   0    FOR SUB-INCREMENT   3
  NUMBER OF ITERATIONS =   1    FOR SUB-INCREMENT   4
  NUMBER OF ITERATIONS =   0    FOR SUB-INCREMENT   5
  NUMBER OF ITERATIONS =   1    FOR SUB-INCREMENT   6
  NUMBER OF ITERATIONS =   0    FOR SUB-INCREMENT   7
```

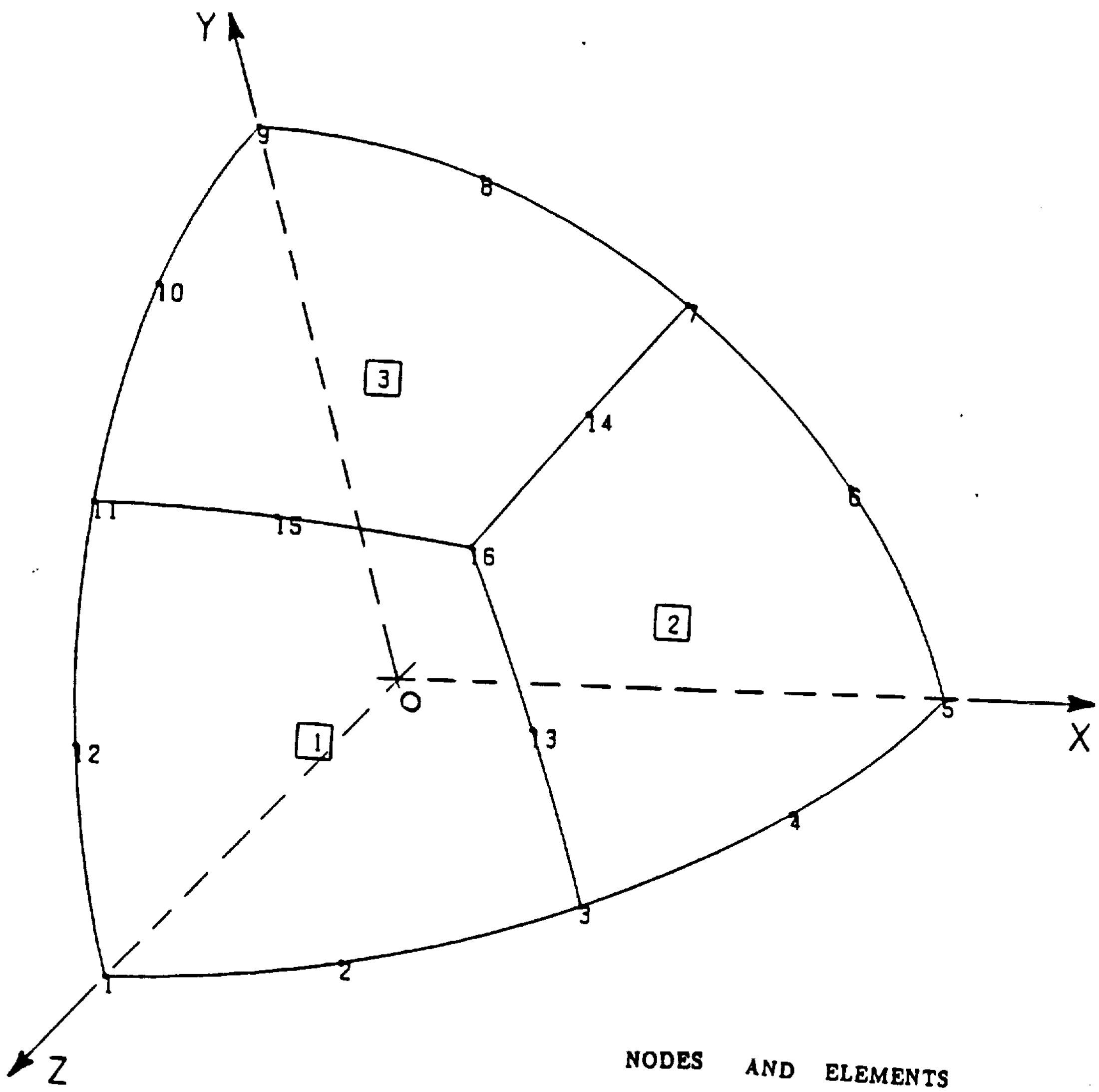
```
JOB TITLE:    CYCLIC LOADING OF UNIT CUBE
BOUNDARY SOLUTION AT TIME =    30.0000       FOR REGION = GNR1
```

NODE	EPS-XX	EPS-YY	EPS-ZZ	EPS-XY	EPS-XZ	EPS-YZ
1	0.189353E-02	0.189436E-02	-0.588885E-02	0.000000E+00	0.000000E+00	0.000000E+00
2	0.189409E-02	0.189337E-02	-0.588816E-02	-0.201241E-07	0.000000E+00	0.352786E-06
3	0.189377E-02	0.189264E-02	-0.588792E-02	-0.144105E-07	0.000000E+00	0.548964E-06
4	0.189326E-02	0.189320E-02	-0.588814E-02	0.337206E-07	0.348383E-06	0.000000E+00
5	0.189313E-02	0.189295E-02	-0.588803E-02	0.294308E-07	0.460641E-06	0.380140E-06
6	0.189342E-02	0.189333E-02	-0.588843E-02	0.290599E-07	0.000000E+00	0.266784E-06
7	0.189295E-02	0.189407E-02	-0.588852E-02	0.312888E-07	0.555922E-06	0.000000E+00
8	0.189312E-02	0.189462E-02	-0.588844E-02	0.461091E-07	0.480914E-06	0.000000E+00
201	0.189331E-02	0.189353E-02	-0.588765E-02	0.000000E+00	-0.409453E-06	-0.327060E-06
203	0.189369E-02	0.189291E-02	-0.588690E-02	0.000000E+00	-0.431805E-06	0.464418E-06
205	0.189294E-02	0.189298E-02	-0.588620E-02	0.000000E+00	0.479054E-06	0.452239E-06
207	0.189273E-02	0.189377E-02	-0.588678E-02	0.000000E+00	0.394132E-06	-0.359418E-06
101	0.189415E-02	0.189377E-02	-0.588728E-02	0.000000E+00	-0.436227E-06	-0.475820E-06
102	0.189427E-02	0.189291E-02	-0.588764E-02	0.125702E-06	0.000000E+00	-0.307565E-06
103	0.189398E-02	0.189238E-02	-0.588668E-02	0.144236E-06	-0.469929E-06	0.999745E-07
104	0.189293E-02	0.189281E-02	-0.588679E-02	-0.152102E-06	-0.584812E-07	0.000000E+00
105	0.189225E-02	0.189242E-02	-0.588544E-02	-0.352706E-06	0.382001E-06	0.205490E-06
106	0.189240E-02	0.189289E-02	-0.588633E-02	-0.801115E-07	0.000000E+00	-0.252025E-06
107	0.189171E-02	0.189372E-02	-0.588578E-02	0.207518E-06	0.149027E-06	-0.537781E-06
108	0.189269E-02	0.189400E-02	-0.588722E-02	0.123943E-06	-0.122994E-06	0.000000E+00

```
JOB TITLE:    CYCLIC LOADING OF UNIT CUBE
INELASTIC SOLUTION AT TIME =    30.0000

NODE TYPE:   E = ELASTIC NODE ,   P = PLASTIC NODE
             ME = METAELASTIC NODE
```

NODE	TYPE	EQUIVALENT PLASTIC STRAIN	EQUIVALENT STRESS
1	ME	6.47921E-04	1.24692E+02
2	ME	6.47766E-04	1.24680E+02
3	ME	6.47637E-04	1.24669E+02
4	ME	6.48865E-04	1.24640E+02
5	ME	6.48929E-04	1.24633E+02
6	ME	6.49108E-04	1.24641E+02
7	ME	6.48107E-04	1.24673E+02
8	ME	6.47982E-04	1.24682E+02
201	ME	6.49063E-04	1.24629E+02
203	ME	6.48976E-04	1.24616E+02
205	ME	6.48733E-04	1.24603E+02
207	ME	6.48905E-04	1.24615E+02
101	ME	6.49319E-04	1.24625E+02
102	ME	6.49366E-04	1.24623E+02
103	ME	6.49369E-04	1.24599E+02
104	ME	6.49122E-04	1.24602E+02
105	ME	6.48701E-04	1.24579E+02
106	ME	6.48796E-04	1.24598E+02
107	ME	6.48706E-04	1.24592E+02
108	ME	6.49066E-04	1.24620E+02

```
EXAMPLE PROBLEM: TRAN601

ANALYSIS TYPE: TRANSIENT ELASTODYNAMICS
  3-D, TRANSIENT ELASTODYNAMICS BY CONVOLUTION INTEGRALS AND
  LINEAR VARIATION.

PROBLEM DESCRIPTION:
  DETERMINE THE TIME-DEPENDENT RADIAL DISPLACEMENT AT THE SURFACE
  OF A SPHERICAL CAVITY(RADIUS=212) IN AN INFINITE MEDIUM (POISSON'S
  RATIO=0.25, YOUNG'S MODULUS=8993000, DENSITY=0.00025) UNDER A
  HEAVISIDE INTERNAL PRESSURE 1000H(T).  INITIAL DISPLACEMENT
  CONDITIONS ARE ZERO.

BOUNDARY ELEMENT MODEL:
  AN OCTANT MODEL OF A SPHERICAL SURFACE USING THREE 8-NODED
  QUADRATIC SURFACE ELEMENTS.  GLOBAL BOUNDARY CONDITIONS
  SPECIFIED EVERYWHERE.

REFERENCE FOR ANALYTICAL SOLUTION:
  TIMOSHENKO, THEORY OF ELASTICITY, PG. 509-513.

SOLUTION POINTS TO VERIFY:
  (DISPLACEMENT AT POINT 5(212,0,0) IN X-DIRECTION)
  TIME STEP      TIME      POINT       BEST SOL      ANALYTICAL SOL.

      1       3.5000E-04     5       .63449E-02       6.2534E-03
      2       7.0000E-04     5       .11018E-01       1.1265E-02
     10       3.5000E-03     5       .15500E-01       1.6127E-02

RUN TIME:

MISCELLANEOUS:
  THIS PROBLEM IS AN EXTERIOR PROBLEM, SO AN EXTERIOR
  CARD IS REQUIRED.  BY UTILIZING SYMMETRY, ONLY ONE
  OCTANT GEOMETRY AND BOUNDARY CONDITIONS ARE REQUIRED.
```



NODES AND ELEMENTS

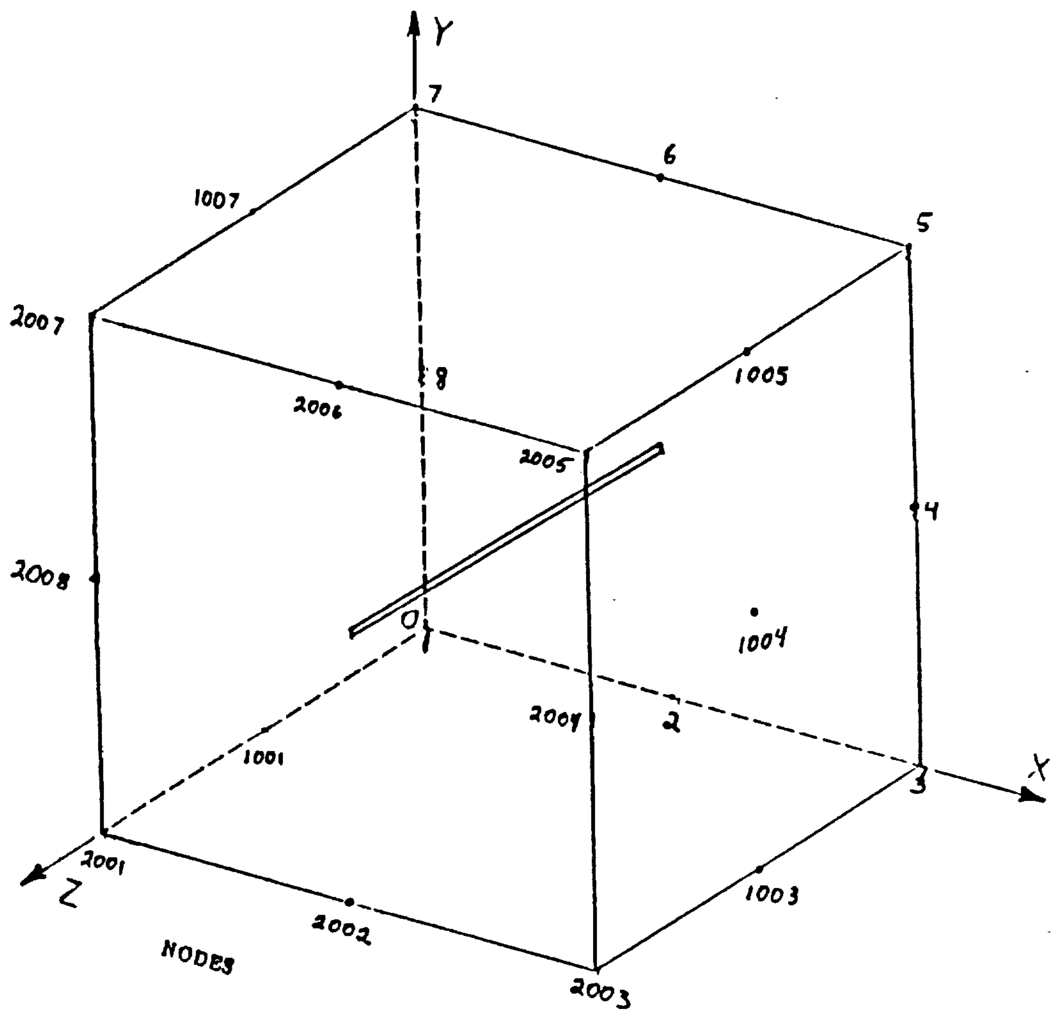
```
**CASE
   TITL A SPHERICAL CAVITY IN INFINITE SPACE UNDER INTERNAL PRESSURE
   TRANSIENT  10  0.00035
   TIME VARIATION LINEAR
   SYMMETRY OCTAN
   PRINT BOUN


**MAT
    ID MAT1
    EMOD 8993000.0
    POISS 0.25
    DENS 0.00025


**GMR
   ID GMR1
   MAT MAT1
   EXTERIOR                          $ NOTE: EXTERIOR PROBLEM
   POINTS
 1     0.000     0.000   212.000
 2    81.129     0.000   195.863
 3   149.907     0.000   149.907
 4   195.863     0.000    81.129
 5   212.000     0.000     0.000
 6   195.863    81.129     0.000
 7   149.907   149.907     0.000
 8    81.129   195.863     0.000
 9     0.000   212.000     0.000
10     0.000   195.863    81.129
11     0.000   149.907   149.907
12     0.000    81.129   195.863
13   142.864    64.216   142.864
14   142.864   142.864    64.216
15    64.216   142.864   142.864
16   122.398   122.398   122.398
  SURFACE SURF1
    TYPE QUAD
    ELEMENTS
       1     1  2  3 13 16 15 11 12
       2     3  4  5  6  7 14 16 13
       3    16 14  7  8  9 10 11 15
   NORMAL  1 -


**BCSET
  ID  BCS1
   VALUE
   GMR GMR1
   SURFACE SURF1
     ELEMENTS  1  2  3
 TRAC 1
 SPLIST 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16
 T 1     0. 382.684 707.109 923.882 1000. 923.882 707.109 382.684 0. 0.
```

```
T 1     0. 0. 673.887 673.887 302.906 577.349
TRAC 2
SPLIST 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16
T 1     0. 0. 0. 0. 0. 382.684 707.109 923.882 1000. 923.882 707.109
T 1     382.684 302.906 673.887 673.887 577.349
TRAC 3
SPLIST 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16
T 1     1000. 923.882 707.109 382.684 0. 0. 0. 0. 0. 382.684 707.109
T 1     923.882 673.887 302.906 673.887 577.349
$
$  END OF DATA
```

JOB TITLE: A SPHERICAL CAVITY IN INFINITE SPACE UNDER INTERNAL PRESSURE
BOUNDARY SOLUTION AT TIME = 0.00035 FOR REGION = GMR1

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	Z-DISPL.	X-TRAC.	Y TRAC.	Z TRAC.
1	1	-0.52759E-19	-0.12662E-18	0.63449E-02	0.00000E+00	0.00000E+00	0.10000E+04
1	12	-0.42061E-18	0.23266E-02	0.56283E-02	0.00000E+00	0.38268E+03	0.92388E+03
1	11	0.17874E-18	0.43596E-02	0.43596E-02	0.00000E+00	0.70711E+03	0.70711E+03
1	15	0.18508E-02	0.41402E-02	0.41402E-02	0.30291E+03	0.67389E+03	0.67389E+03
1	16	0.35235E-02	0.35235E-02	0.35235E-02	0.57735E+03	0.57735E+03	0.57735E+03
1	13	0.41402E-02	0.18508E-02	0.41402E-02	0.67389E+03	0.30291E+03	0.67389E+03
1	3	0.43596E-02	-0.37006E-18	0.43596E-02	0.70711E+03	0.00000E+00	0.70711E+03
1	2	0.23266E-02	0.18577E-18	0.56283E-02	0.38268E+03	0.00000E+00	0.92388E+03
2	3	0.43596E-02	-0.37006E-18	0.43596E-02	0.70711E+03	0.00000E+00	0.70711E+03
2	13	0.41402E-02	0.18508E-02	0.41402E-02	0.67389E+03	0.30291E+03	0.67389E+03
2	16	0.35235E-02	0.35235E-02	0.35235E-02	0.57735E+03	0.57735E+03	0.57735E+03
2	14	0.41402E-02	0.41402E-02	0.18508E-02	0.67389E+03	0.67389E+03	0.30291E+03
2	7	0.43596E-02	0.43596E-02	-0.15514E-18	0.70711E+03	0.70711E+03	0.00000E+00
2	6	0.56283E-02	0.23266E-02	-0.60646E-18	0.92388E+03	0.38268E+03	0.00000E+00
2	5	0.63449E-02	0.19255E-18	0.58961E-19	0.10000E+04	0.00000E+00	0.00000E+00
2	4	0.56283E-02	-0.22963E-18	0.23266E-02	0.92388E+03	0.00000E+00	0.38268E+03
3	16	0.35235E-02	0.35235E-02	0.35235E-02	0.57735E+03	0.57735E+03	0.57735E+03
3	15	0.18508E-02	0.41402E-02	0.41402E-02	0.30291E+03	0.67389E+03	0.67389E+03
3	11	0.17874E-18	0.43596E-02	0.43596E-02	0.00000E+00	0.70711E+03	0.70711E+03
3	10	-0.53732E-20	0.56283E-02	0.23266E-02	0.00000E+00	0.92388E+03	0.38268E+03
3	9	0.90690E-19	0.63449E-02	-0.56866E-18	0.00000E+00	0.10000E+04	0.00000E+00
3	8	0.23266E-02	0.56283E-02	0.28133E-18	0.38268E+03	0.92388E+03	0.00000E+00
3	7	0.43596E-02	0.43596E-02	-0.15514E-18	0.70711E+03	0.70711E+03	0.00000E+00
3	14	0.41402E-02	0.41402E-02	0.18508E-02	0.67389E+03	0.67389E+03	0.30291E+03

```
VERIFICATION PROBLEM:  D.403

ANALYSIS TYPE:
  ELASTIC ANALYSIS OF A 3-D CUBE WITH HOLES (USING HOLE ELEMENTS)

PROBLEM DISCRIPTION:
  A 3-D UNIT CUBE HAS A TUBULAR HOLE OF RADIUS .1 CENTERED IN THE
  MIDDLE RUNNING FROM ONE SIDE TO THE OTHER.  THE HOLE IS SUBJECTED
  TO A UNIFORM PRESSURE APPLIED IN THE HOLE.

BOUNDARY ELEMENT MODEL:
  3-D UNIT CUBE IS MODELED WITH SIX QUADRATIC ELEMENTS AND THE HOLE
  IS MODELED WITH A SINGLE QUADRATIC LINE ELEMENT.

REFERENCE FOR SOLUTION:
  RESULTS FROM EXACT BEM MODELING OF THE HOLE USING SURFACE PATCHES
  ARE USED FOR COMPARISION.

SOLUTION POINTS TO VERIFY:

  TIME 1:

     NODE          X-DISPLACEMENT
     1004            0.06607

RUN TIME:
    0.73 X BASE PROBLEM

MISCELLANEOUS:
  NONE
```


Y
7
6
5
1007
2007
8
1005
2006
2005
4
2008
0
1004
1
2004
2
X
1001
3
2001
Z
1003
2002
NODES
2003

```
**CASE
  TITLE  - D.403 -   3-D CUBE WITH HOLE SUBJECTED TO INTERNAL PRESSURE
    TIMES  1.
    ECHO

**MATE
  ID MAT1
  TEMP 70.0
  EMOD  100.
  POIS  0.3
  DENSITY  1.
  ALPHA  1.

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
         1         .0000         .0000         .0000
         2         .5000         .0000         .0000
         3        1.0000         .0000         .0000
         4        1.0000         .5000         .0000
         5        1.0000        1.0000         .0000
         6         .5000        1.0000         .0000
         7         .0000        1.0000         .0000
         8         .0000         .5000         .0000
      1001         .0000         .0000         .5000
      1003        1.0000         .0000         .5000
      1004        1.0000         .5000         .5000
      1005        1.0000        1.0000         .5000
      1007         .0000        1.0000         .5000
      2001         .0000         .0000        1.0000
      2002         .5000         .0000        1.0000
      2003        1.0000         .0000        1.0000
      2004        1.0000         .5000        1.0000
      2005        1.0000        1.0000        1.0000
      2006         .5000        1.0000        1.0000
      2007         .0000        1.0000        1.0000
      2008         .0000         .5000        1.0000
  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
     101         1     2     3  1003  2003  2002  2001  1001
     102         3     4     5  1005  2005  2004  2003  1003 1004
     103         5     6     7  1007  2007  2006  2005  1005
     104         7     8     1  1001  2001  2008  2007  1007
       1         1     2     3     4     5     6     7     8
     201      2001  2002  2003  2004  2005  2006  2007  2008
    NORMAL  201 +
  HOLE
    POINTS
     3001 .5 .5 0.0
     3005 .5 .5 0.5
     3009 .5 .5 1.0
    TYPE QUAD
```

```
    ELEMENTS
      301   .1   3001   3005   3009

**BCSET
  ID  BC1
  VALUE
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   104
  DISP  1
    SPLIST ALL
      T  1   0.0

**BCSET
  ID  BC2
  VALUE
  GMR  GMR1
    SURFACE  SURF11
    ELEMENTS   104
    POINTS 1001 1007
      DISP  3
      SPLIST 1001 1007
        T  1   0.0 0.0

**BCSET
  ID  BC3
  VALUE
  GMR  GMR1
    SURFACE  SURF11
    ELEMENTS   104
    POINTS  8 2008
      DISP  2
      SPLIST 8 2008
        T  1   0.0 0.0

**BCSET
  ID  PRESSURE
  VALUE
  GMR  GMR1
    HOLE
    ELEMENT   301
      PRESS
      SPLIST ALL
        T  1  100.

$ END OF DATA
```

VERIFICATION PROBLEM: D.404

ANALYSIS TYPE:
THERMOELASTIC ANALYSIS OF A 3-D CUBE WITH A HOLE

PROBLEM DISCRIPTION:
A 3-D UNIT CUBE HAS A TUBULAR HOLE OF RADIUS .1 CENTERED IN THE MIDDLE RUNNING FROM ONE SIDE TO THE OTHER. THE CUBE IS SUBJECTED TO A UNIFORM TRACTION APPLIED ON A FACE OF THE CUBE PERDICULAR TO THE HOLE.

BOUNDARY ELEMENT MODEL:
3-D UNIT CUBE IS MODELED WITH SIX QUADRATIC ELEMENTS AND THE HOLE IS MODELED WITH THREE QUADRATIC LINE ELEMENTS.

REFERENCE FOR ANALYTICAL SOLUTION:
BOLEY AND WEINNER (1960), THEORY OF THERMAL STRESSES.
SOLUTION FOR FREE EXPANSION OF A UNRESTRAINED BODY UNDER UNIFORM AND LINEAR TEMPERATURE DISTRIBUTIONS

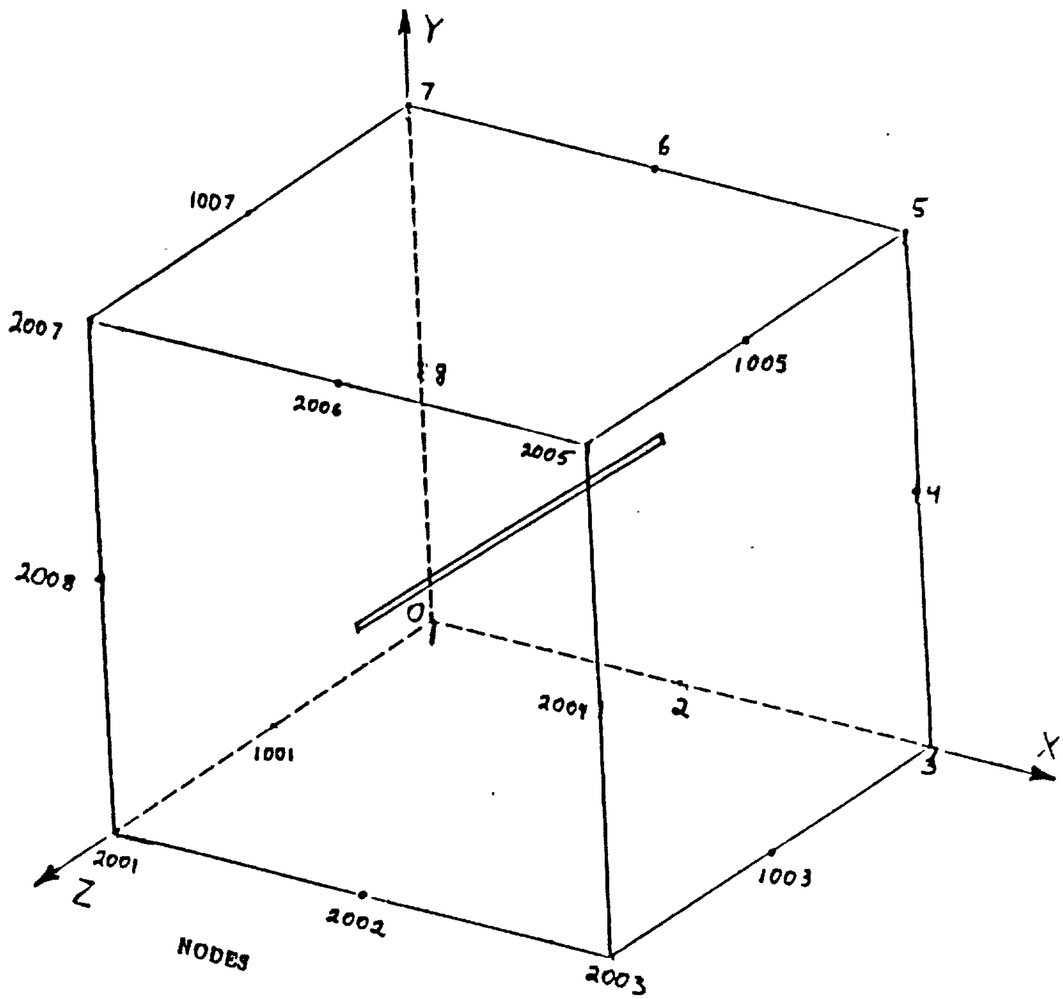
SOLUTION POINTS TO VERIFY:

TIME 1:

NODE	X-DISPLACEMENT ANAL.	X-DISPLACEMENT BEST	Y-DISPLACEMENT ANAL.	Y-DISPLACEMENT BEST
4	1.0	.999	.0	.000
5	1.0	0.996	.5	.499
5001	.5	0.497	.2	.199

RUN TIME:
1.54 X BASE PROBLEM

MISCELLANEOUS:
NONE


Y
7
6
5
1007
2007
1005
8
2006
2005
4
2008
0
1
2004
2
1001
3
X
2001
Z
1003
2002
NODES
2003

```
**CASE
  TITLE  - D.404 -  3-D CUBE WITH HOLE SUBJECTED TO A THERMAL BODY FORCE
    TIMES  1.
    ECHO

**MATE
  ID MAT1
  TEMP 70.0
  EMOD  100.
  POIS  0.
  ALPHA  1.

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  TINI 0.0
  POINTS
         1        .0000       .0000       .0000
         2        .5000       .0000       .0000
         3       1.0000       .0000       .0000
         4       1.0000       .5000       .0000
         5       1.0000      1.0000       .0000
         6        .5000      1.0000       .0000
         7        .0000      1.0000       .0000
         8        .0000       .5000       .0000
      1001        .0000       .0000      1.0000
      1003       1.0000       .0000      1.0000
      1005       1.0000      1.0000      1.0000
      1007        .0000      1.0000      1.0000
      2001        .0000       .0000      2.0000
      2002        .5000       .0000      2.0000
      2003       1.0000       .0000      2.0000
      2004       1.0000       .5000      2.0000
      2005       1.0000      1.0000      2.0000
      2006        .5000      1.0000      2.0000
      2007        .0000      1.0000      2.0000
      2008        .0000       .5000      2.0000
  SURFACE SURF11
    TYPE QUAD
    ELEMENTS
     101        1     2     3  1003  2003  2002  2001  1001
     102        3     4     5  1005  2005  2004  2003  1003
     103        5     6     7  1007  2007  2006  2005  1005
     104        7     8     1  1001  2001  2008  2007  1007
       1        1     2     3     4     5     6     7     8
     201     2001  2002  2003  2004  2005  2006  2007  2008
    NORMAL  201 +
  HOLE
    POINTS
     3001 .5 .5 0.
     3002 .5 .5 0.25
     3003 .5 .5 0.5
     3004 .5 .5 1.0
     3005 .5 .5 1.5
```

```
     3006 .5 .5 1.75
     3007 .5 .5 2.0
    TYPE QUAD
    ELEMENTS
      301  .1  3001  3002  3003
      302  .1  3003  3004  3005
      303  .1  3005  3006  3007
    SAMPLING POINTS
     5001 .5  .7 1.
   PART INT.
   NODE
     5001

**BCSET
  ID  BC1
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   104
  DISP  1
    SPLIST ALL
      T  1   0.0

**BCSET
  ID  BC2
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   104
    POINTS 1001 1007
  DISP  3
    SPLIST 1001 1007
      T  1   0. 0.

**BCSET
  ID  BC3
  GMR  GMR1
  SURFACE  SURF11
    ELEMENTS   104
    POINTS  8 2008
  DISP  2
    SPLIST 8 2008
      T  1   0. 0.

**BODY FORCE
  THERMAL
  TIME 1.
    GMR GMR1
    TEMPS
      5001   1.

$ END OF DATA
```

EXAMPLE PROBLEM: ELAS603

ANALYSIS TYPE:
ELASTICITY (USING HOLE ELEMENTS)
3-D, STATIC, ELASTIC ANALYSIS OF A BODY WITH HOLES

PROBLEM DESCRIPTION:
A 3-D, FIXED END THICK CYLINDER WITH HOLES IN THE CIRCUMFERENTIAL DIRECTION IS ANALYZED SUBJECTED TO INTERNAL PRESSURE IN THE CYLINDER.

BOUNDARY ELEMENT MODEL:
THE INNER AND OUTER RADII OF THE CYLINDER ARE 10 AND 20, RESPECTIVELY, THE THICKNESS IS 2 AND THE RADII OF THE HOLES ARE 0.5. BY USING ROLLER BOUNDARY CONDITIONS ON THE FACES OF SYMMETRY, ONLY A FIFTEEN DEGREE SLICE OF THE THICK CYLINDER NEEDS TO BE MODELED. SIXTEEN EIGHT-NODED QUADRATIC BOUNDARY ELEMENTS ARE USED TO DEFINE THE SIDES OF THE MODEL. A NINE-NODED ELEMENT IS USED ON BOTH THE INTERNAL AND EXTERNAL FACES OF THE CYLINDER, AND ONE HOLE ELEMENT IS USED PER HOLE. NOTE THE HOLE ELEMENTS ARE CURVILINEAR IN GEOMETRY.

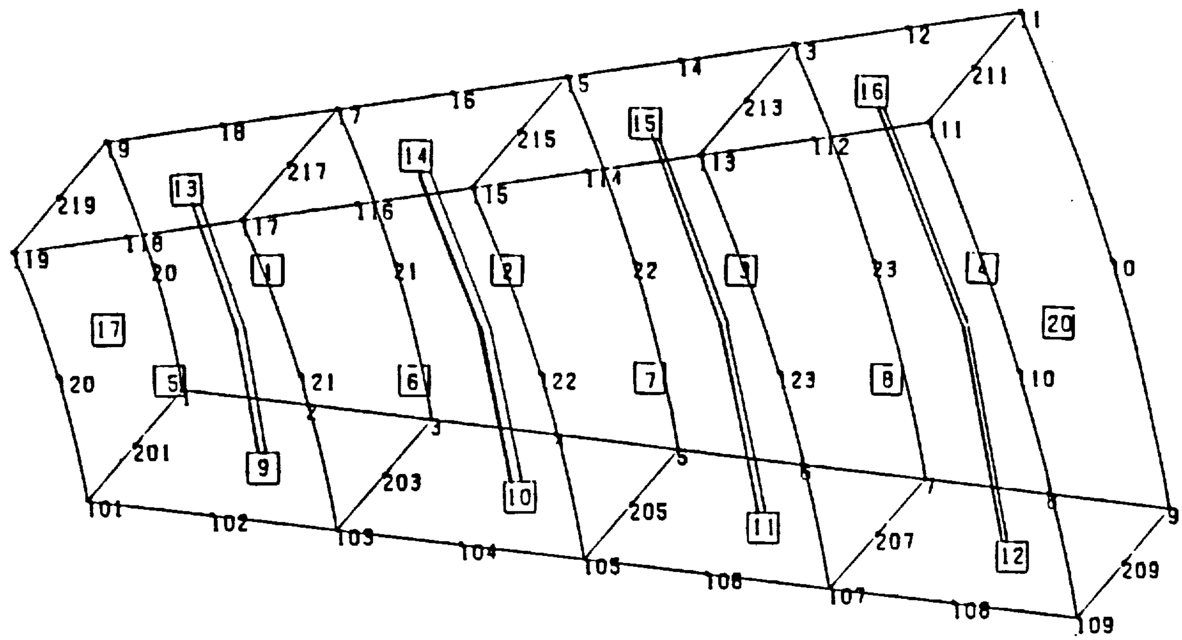
REFERENCE FOR ANALYTICAL SOLUTION:
RESULTS FROM A AXISYMMETRIC BEM ANALYSIS ARE USED FOR COMPARISON WITH THE 3-D HOLE RESULTS FROM THE PRESENT DATA.

SOLUTION POINTS TO VERIFY:

NODE	RADIAL POSITION	RADIAL X-DISPLACEMENT	CIRCUMFERENTIAL YY-STRESS
20	10.0	.077708	223.5E+03
21	12.5	.064537	153.5E+03
22	15.0	.056174	116.6E+03
23	17.5	.050939	94.9E+03
10	20.0	.047734	81.0E+03

RUN TIME:
11 X BASE PROBLEM

MISCELLANEOUS:



NODES AND ELEMENTS

```
**CASE CONTROL
  TITL THICK CYLINDER WITH HOLES UNDER INTERNAL PRESSURE (SINGLE REGION)

**MATERIAL INPUT
  ID MAT1
  TEMP 70.0
  EMOD 30.0E+06
  POIS 0.3


**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  POINTS
    1      9.9144    -1.3053    0.0
    2     11.1538    -1.4684    0.0
    3     12.3931    -1.6316    0.0
    4     13.6324    -1.7947    0.0
    5     14.8717    -1.9579    0.0
    6     16.1110    -2.1211    0.0
    7     17.3503    -2.2842    0.0
    8     18.5896    -2.4474    0.0
    9     19.8289    -2.6105    0.0
   10     20.0        0.0       0.0
   11     19.8289     2.6105    0.0
   12     18.5896     2.4474    0.0
   13     17.3503     2.2842    0.0
   14     16.1110     2.1211    0.0
   15     14.8717     1.9579    0.0
   16     13.6324     1.7947    0.0
   17     12.3931     1.6316    0.0
   18     11.1538     1.4684    0.0
   19      9.9144     1.3053    0.0
   20     10.0        0.0       0.0
   21     12.5        0.0       0.0
   22     15.0        0.0       0.0
   23     17.5        0.0       0.0
  101      9.9144    -1.3053    2.0
  102     11.1538    -1.4684    2.0
  103     12.3931    -1.6316    2.0
  104     13.6324    -1.7947    2.0
  105     14.8717    -1.9579    2.0
  106     16.1110    -2.1211    2.0
  107     17.3503    -2.2842    2.0
  108     18.5896    -2.4474    2.0
  109     19.8289    -2.6105    2.0
  110     20.0        0.0       2.0
  111     19.8289     2.6105    2.0
  112     18.5896     2.4474    2.0
  113     17.3503     2.2842    2.0
  114     16.1110     2.1211    2.0
  115     14.8717     1.9579    2.0
```

```
     116      13.6324      1.7947     2.0
     117      12.3931      1.6316     2.0
     118      11.1538      1.4684     2.0
     119       9.9144      1.3053     2.0
     120      10.0         0.0        2.0
     121      12.5         0.0        2.0
     122      15.0         0.0        2.0
     123      17.5         0.0        2.0
     201       9.9144     -1.3053     1.0
     203      12.3931     -1.6316     1.0
     205      14.8717     -1.9579     1.0
     207      17.3503     -2.2842     1.0
     209      19.8289     -2.6105     1.0
     210      20.0         0.0        1.0
     211      19.8289      2.6105     1.0
     213      17.3503      2.2842     1.0
     215      14.8717      1.9579     1.0
     217      12.3931      1.6316     1.0
     219       9.9144      1.3053     1.0
     220      10.0         0.0        1.0
$
$  DEFINE SURFACE BOUNDARY ELEMENTS
$
 SURFACE SURF1
    TYPE QUAD
    ELEMENTS
  1      1   20   19   18   17   21    3    2
  2      3   21   17   16   15   22    5    4
  3      5   22   15   14   13   23    7    6
  4      7   23   13   12   11   10    9    8
  5    101  102  103  121  117  118  119  120
  6    103  104  105  122  115  116  117  121
  7    105  106  107  123  113  114  115  122
  8    107  108  109  110  111  112  113  123
  9    101  201    1    2    3  203  103  102
 10    103  203    3    4    5  205  105  104
 11    105  205    5    6    7  207  107  106
 12    107  207    7    8    9  209  109  108
 13     19  219  119  118  117  217   17   18
 14     17  217  117  116  115  215   15   16
 15     15  215  115  114  113  213   13   14
 16     13  213  113  112  111  211   11   12
 17      1  201  101  120  119  219   19   20  220
 20    109  209    9   10   11  211  111  110  210
  NORMAL 1 +
$
$  DEFINE HOLE POINTS AND ELEMENTS
$
 HOLE
  POINTS
   401  11.1538  1.4684  1.
   402  13.6324  1.7947  1.
   403  16.1110  2.1211  1.
   404  18.5896  2.4474  1.
   501  11.25  0.  1.
   502  13.75  0.  1.
   503  16.25  0.  1.
   504  18.75  0.  1.
```

```
 601  11.1538 -1.4684  1.
 602  13.6324 -1.7947  1.
 603  16.1110 -2.1211  1.
 604  18.5896 -2.4474  1.
  TYPE QUAD
  ELEMENTS
701 .5  401 501 601
702 .5  402 502 602
703 .5  403 503 603
704 .5  404 504 604


$
$  RESTRICT DISPLACEMENT IN THE Z DIRECTION
$
**BCSET
  ID DISP1
  VALUE
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 1 2 3 4 5 6 7 8
    DISP 3
    SPLIST ALL
       T 1 0.0

$
$  ROLLER B.C. ON SYMMETRICAL FACES (SPECIFIED IN THE LOCAL SYSTEM)
$
**BCSET
  ID DISP2
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 9 10 11 12 13 14 15 16
    DISP 1
    SPLIST ALL
       T 1 0.0

$
$  INTERNAL PRESSURE (SPECIFIED IN THE LOCAL SYSTEM)
$
**BCSET
  ID TRAC12
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 17
    TRAC 1
    SPLIST ALL
      T  1 -100000.
$
$  END OF DATA
```

1 JOB TITLE: THICK CYLINDER WITH HOLES UNDER INTERNAL PRESSURE (SINGLE REGION)
HOLE ELEMENT SOLUTION AT TIME = 0.0000 FOR REGION = GMR1

ELEMENT	NODE NO.	X-DISPL.	Y-DISPL.	Z-DISPL.	PRESSURE
701	70401	0.68357E-01	0.99821E-02	0.76308E-09	0.00000E+00
701	80401	0.70931E-01	0.10212E-01	0.14288E-03	0.00000E+00
701	90401	0.70931E-01	0.10212E-01	-0.14288E-03	0.00000E+00
701	70501	0.69165E-01	0.15084E-06	-0.78924E-10	0.00000E+00
701	80501	0.71396E-01	0.52787E-06	0.48832E-03	0.00000E+00
701	90501	0.71396E-01	0.53242E-06	-0.48832E-03	0.00000E+00
701	70601	0.68357E-01	-0.99840E-02	-0.82003E-09	0.00000E+00
701	80601	0.70934E-01	-0.10215E-01	0.14294E-03	0.00000E+00
701	90601	0.70934E-01	-0.10215E-01	-0.14294E-03	0.00000E+00

JOB TITLE: THICK CYLINDER WITH HOLES UNDER INTERNAL PRESSURE (SINGLE REGION)
NODAL OUTPUT AT TIME = 0.000000 FOR REGION = GMR1

NODE	DISPLACEMENT	--------- STRESS ---------		--------- STRAIN ---------	
	X/Y/Z	XX/YY/ZZ	XY/XZ/YZ	XX/YY/ZZ	XY/XZ/YZ
19	0.76942E-01	-0.87392E+05	-0.42497E+05	-0.57518E-02	-0.18415E-02
	0.10122E-01	0.22617E+06	0.82712E+04	0.78359E-02	0.35842E-03
	0.00000E+00	0.57699E+05	0.10913E+04	0.53553E-03	0.47290E-04
20	0.77708E-01	-0.93024E+05	-0.18940E+01	-0.59620E-02	-0.82072E-07
	-0.93412E-07	0.22351E+06	0.00000E+00	0.77544E-02	0.00000E+00
	0.00000E+00	0.62611E+05	-0.29442E-01	0.78220E-03	-0.12758E-08
21	0.64537E-01	-0.63572E+05	-0.16886E+00	-0.42436E-02	-0.73175E-08
	0.37604E-07	0.15349E+06	0.00000E+00	0.51624E-02	0.00000E+00
	0.00000E+00	0.58967E+05	0.00000E+00	0.10664E-02	0.00000E+00
22	0.56174E-01	-0.29168E+05	-0.16664E+00	-0.25694E-02	-0.72212E-08
	-0.27238E-07	0.11663E+06	0.00000E+00	0.37486E-02	0.00000E+00
	0.00000E+00	0.43082E+05	0.00000E+00	0.56143E-03	0.00000E+00
23	0.50939E-01	-0.98318E+04	-0.78835E+00	-0.16289E-02	-0.34162E-07
	-0.18330E-08	0.94951E+05	0.00000E+00	0.29116E-02	0.00000E+00
	0.00000E+00	0.35170E+05	0.00000E+00	0.32116E-03	0.00000E+00

VERIFICATION PROBLEM: D.406

ANALYSIS TYPE:
ELASTIC-INSERT PROBLEM (USING INSERT ELEMENTS)
3-D, STATIC, ELASTIC ANALYSIS OF A BODY WITH INSERTS

PROBLEM DISCRIPTION:
A 3-D, FIXED END THICK CYLINDER WITH INSERTS IN THE CIRCUMFERENTIAL DIRECTION IS ANALYZED SUBJECTED TO INTERNAL PRESSURE IN THE CYLINDER.

BOUNDARY ELEMENT MODEL:
THE INNER AND OUTER RADII OF THE CYLINDER ARE 10 AND 20, RESPECTIVELY, THE THICKNESS IS 2 AND THE RADII OF THE INSERTS ARE 0.5. BY USING ROLLER BOUNDARY CONDITIONS ON THE FACES OF SYMMETRY, ONLY A FIFTEEN DEGREE SLICE OF THE THICK CYLINDER NEEDS TO BE MODELED. SIXTEEN EIGHT-NODED QUADRATIC BOUNDARY ELEMENTS ARE USED TO DEFINE THE SIDES OF THE MODEL. A NINE-NODED ELEMENT IS USED ON BOTH THE INTERNAL AND EXTERNAL FACES OF THE CYLINDER, AND ONE INSERT ELEMENT IS USED PER INSERT. NOTE THE INSERT ELEMENTS ARE CURVILINEAR IN GEOMETRY.

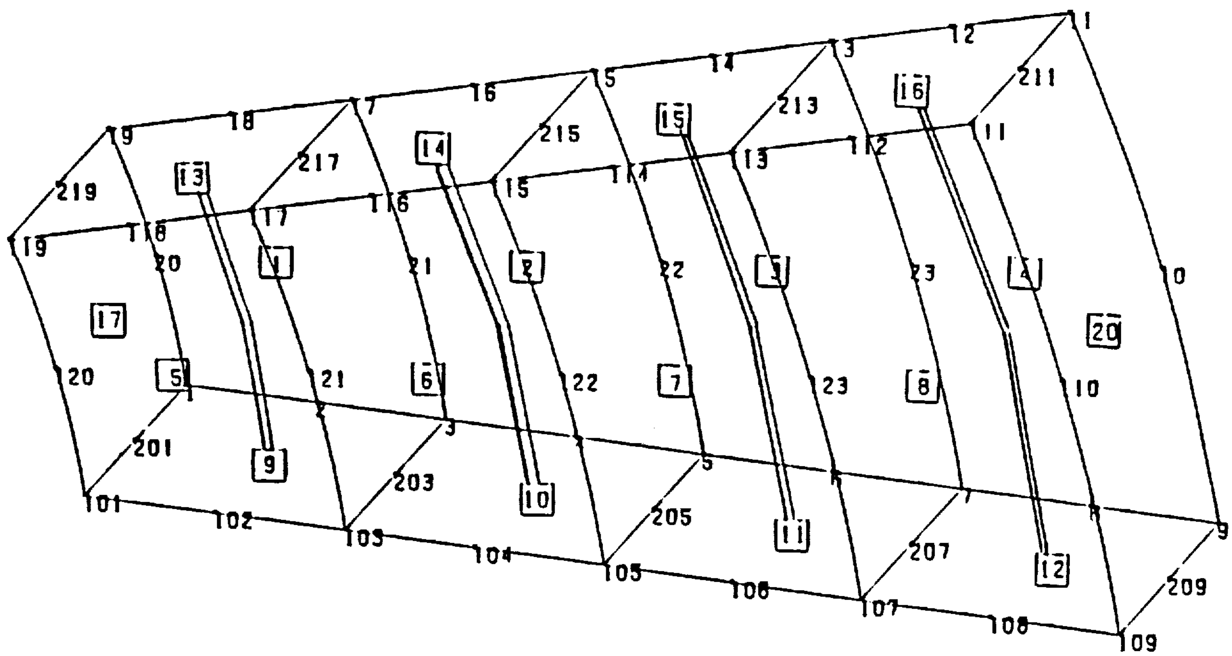
REFERENCE FOR SOLUTION:
RESULTS FROM A MULTI-REGION, AXISYMMETRIC BEM ANALYSIS ARE USED FOR COMPARISON WITH THE 3-D INSERT RESULTS FROM THE PRESENT DATA. 'DEVELOPMENT OF BEM FOR CERAMIC COMPOSITES', FIRST ANNUAL REPORT, NASA (LEWIS RESEARCH CENTER) - GRANT NUMBER: NAG3-888 (DEC. 1988).

SOLUTION POINTS TO VERIFY:

NODE	RADIAL POSITION	RADIAL X-DISPLACEMENT AXIS.	RADIAL X-DISPLACEMENT BEST	CIRCUMFERENTIAL YY-STRESS AXIS.	CIRCUMFERENTIAL YY-STRESS BEST
9020	10.0	.913	.97722	5.11	5.5404
9021	12.5	.712	.80990	4.28	3.7734
9022	15.0	.602	.71539	3.46	3.1350
9023	17.5	.537	.66136	3.04	2.9225
9024	20.0	.500	.63550	2.72	2.8191

RUN TIME:
6.50 X BASE PROBLEM

MISCELLANEOUS:
NONE



NODES AND ELEMENTS

```
**CASE CONTROL
  TITL - D.406 - THICK CYLINDER W/ INSERTS UNDER INTERNAL PRESSURE
  PRINT BOUND
  PRINT LOAD
  ECHO

**MATERIAL INPUT
  ID MAT1
  TEMP 70.0
  EMOD 100.0
  POIS 0.3

**GMR
  ID GMR1
  MAT MAT1
  TREF 70.0
  POINTS
    1      9.9144   -1.3053    0.0
    2     11.1538   -1.4684    0.0
    3     12.3931   -1.6316    0.0
    4     13.6324   -1.7947    0.0
    5     14.8717   -1.9579    0.0
    6     16.1110   -2.1211    0.0
    7     17.3503   -2.2842    0.0
    8     18.5896   -2.4474    0.0
    9     19.8289   -2.6105    0.0
   10     20.0       0.0       0.0
   11     19.8289    2.6105    0.0
   12     18.5896    2.4474    0.0
   13     17.3503    2.2842    0.0
   14     16.1110    2.1211    0.0
   15     14.8717    1.9579    0.0
   16     13.6324    1.7947    0.0
   17     12.3931    1.6316    0.0
   18     11.1538    1.4684    0.0
   19      9.9144    1.3053    0.0
   20     10.0       0.0       0.0
   21     12.5       0.0       0.0
   22     15.0       0.0       0.0
   23     17.5       0.0       0.0
  101      9.9144   -1.3053    2.0
  102     11.1538   -1.4684    2.0
  103     12.3931   -1.6316    2.0
  104     13.6324   -1.7947    2.0
  105     14.8717   -1.9579    2.0
  106     16.1110   -2.1211    2.0
  107     17.3503   -2.2842    2.0
  108     18.5896   -2.4474    2.0
  109     19.8289   -2.6105    2.0
  110     20.0       0.0       2.0
  111     19.8289    2.6105    2.0
  112     18.5896    2.4474    2.0
  113     17.3503    2.2842    2.0
  114     16.1110    2.1211    2.0
  115     14.8717    1.9579    2.0
```

```
 116     13.6324     1.7947    2.0
 117     12.3931     1.6316    2.0
 118     11.1538     1.4684    2.0
 119      9.9144     1.3053    2.0
 120     10.0        0.0       2.0
 121     12.5        0.0       2.0
 122     15.0        0.0       2.0
 123     17.5        0.0       2.0
 201      9.9144    -1.3053    1.0
 203     12.3931    -1.6316    1.0
 205     14.8717    -1.9579    1.0
 207     17.3503    -2.2842    1.0
 209     19.8289    -2.6105    1.0
 210     20.0        0.0       1.0
 211     19.8289     2.6105    1.0
 213     17.3503     2.2842    1.0
 215     14.8717     1.9579    1.0
 217     12.3931     1.6316    1.0
 219      9.9144     1.3053    1.0
 220     10.0        0.0       1.0
$
$  DEFINE SURFACE BOUNDARY ELEMENTS
$
 SURFACE SURF1
   TYPE QUAD
   ELEMENTS
  1     1   20   19   18   17   21    3    2
  2     3   21   17   16   15   22    5    4
  3     5   22   15   14   13   23    7    6
  4     7   23   13   12   11   10    9    8
  5   101  102  103  121  117  118  119  120
  6   103  104  105  122  115  116  117  121
  7   105  106  107  123  113  114  115  122
  8   107  108  109  110  111  112  113  123
  9   101  201    1    2    3  203  103  102
 10   103  203    3    4    5  205  105  104
 11   105  205    5    6    7  207  107  106
 12   107  207    7    8    9  209  109  108
 13    19  219  119  118  117  217   17   18
 14    17  217  117  116  115  215   15   16
 15    15  215  115  114  113  213   13   14
 16    13  213  113  112  111  211   11   12
 17     1  201  101  120  119  219   19   20 220
 20   109  209    9   10   11  211  111  110 210
  NORMAL 1 +
$
$  DEFINE INSERT POINTS AND ELEMENTS
$
 INSERT 1000.0
  POINTS
   401   11.1538   1.4684  1.
   402   13.6324   1.7947  1.
   403   16.1110   2.1211  1.
   404   18.5896   2.4474  1.
   501   11.25   0.  1.
   502   13.75   0.  1.
   503   16.25   0.  1.
   504   18.75   0.  1.
```

```
    601  11.1538 -1.4684  1.
    602  13.6324 -1.7947  1.
    603  16.1110 -2.1211  1.
    604  18.5896 -2.4474  1.
     TYPE QUAD
     ELEMENTS
   701 .5  401 501 601
     ELEMENTS
   702 .5  402 502 602
     ELEMENTS
   703 .5  403 503 603
     ELEMENTS
   704 .5  404 504 604
SAMPLING POINTS
 9020      10.0         0.0        0.0
 9021      12.5         0.0        0.0
 9022      15.0         0.0        0.0
 9023      17.5         0.0        0.0
 9024      20.0         0.0        0.0

$  RESTRICT DISPLACEMENT IN THE Z DIRECTION

**BCSET
  ID DISP1
  VALUE
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 1 2 3 4 5 6 7 8
    DISP 3
    SPLIST ALL
       T 1 0.0

$  ROLLER B.C. ON SYMMETRICAL FACES (SPECIFIED IN THE LOCAL SYSTEM)

**BCSET
  ID DISP2
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 9 10 11 12 13 14 15 16
    DISP 1
    SPLIST ALL
       T 1 0.0

$  INTERNAL PRESSURE (SPECIFIED IN THE LOCAL SYSTEM)

**BCSET
  ID TRAC12
  VALUE
  LOCAL
  GMR GMR1
  SURFACE SURF1
  ELEMENTS 17
    TRAC 1
    SPLIST ALL
      T  1 -10.
```

7.0 APPLICATIONS

The analyses developed in **BEST3D** have been applied to a vast array of realistic engineering problems in references listed in Section 8. A few representative examples are described in the following sections:

7.1 Elastic Analysis

Multi-region cylinder
A column attached to an Elastic Half space
Turbine Component Analysis
A Cube with a Hole
Analysis of a thick cylinder with four Holes
Cube with a single Insert
Lateral Behavior of a Cube with Multiple Inserts
Thick Cylinder with Circular Insert Supports
Cube with Multiple Inserts with Random Orientation
Beam with Insert Reinforcement in Bending
Laminated Fiber Composite

7.2 Vibration Analysis

Free Vibration of a Rectangular parallelpiped
Calculation of Natural Frequency of Twisted Plates
Comparison with Finite Element Results for an Automotive component

7.3 Heat transfer Analysis

Tube and Disk Fin Heat Exchanger
Heat Transfer with Hole Elements
Cooling of a Steel Sphere
Injection Mold
Thermal Response of Turbine Blade

7.4 Dynamic Analysis

Dynamic Expansion of a Spherical Cavity
A Column subject to a Transient End Load
Dynamic Analysis of a buried Foundation

Application to the Vibration Isolation Problems

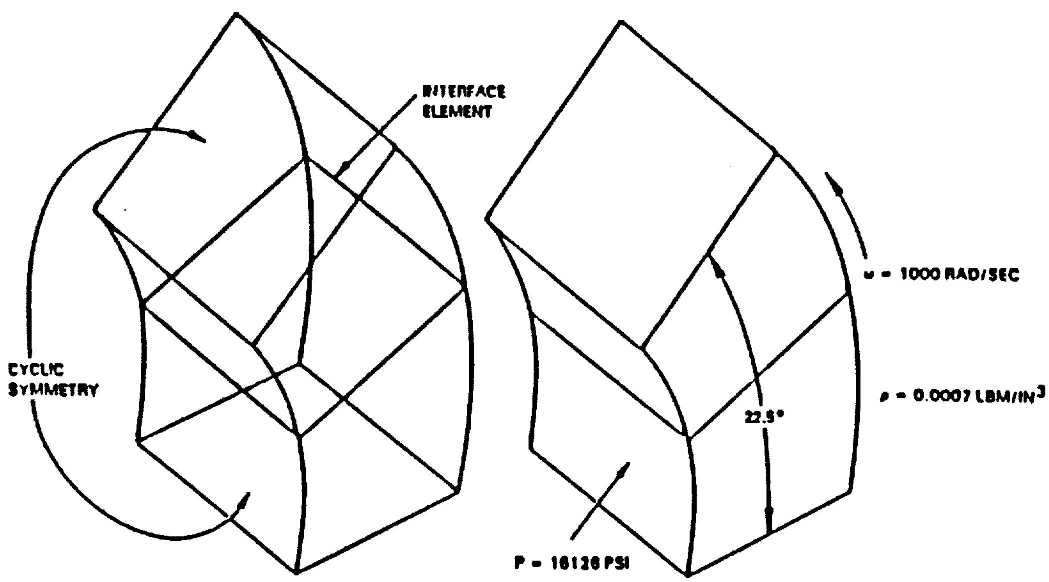
7.5 Non-Linear Stress Analysis

Benchmark Notch Specimen
Three-dimensional Analysis of a Notch Plate
Three-dimensional Analysis of a Perforated Plate

7.1 ELASTIC ANALYSIS

Multi-region Cylinder [33]:

In this problem a 22.5 degree sector of a thick cylinder was analyzed. In this case, several features are present. First, only twelve elements are used. Second, the structure is broken into two regions along the 11.25 degree plane. Third, the boundary conditions on the 0 and 22.5 degree planes have a cyclic symmetry between these two surfaces, rather than the roller boundary conditions. In this case only the results for quadratic variation of displacements and tractions are shown. Good agreement between the calculated and analytical results was obtained (Figure 7.1.1).



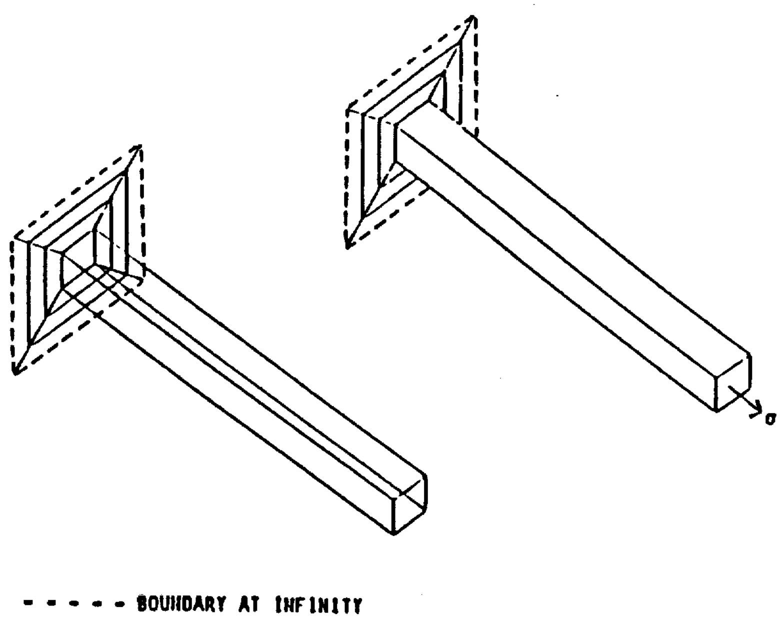
		PRESSURE		SPEED	
		ANALYTICAL	BEST	ANALYTICAL	BEST
ID	U_R	6.90×10^{-3}	6.91×10^{-3}	7.13×10^{-4}	7.13×10^{-4}
	σ_R	-16126	-16126	21	0
	σ_θ	41926	41997	5752	5751
OD	U_R	4.80×10^{-3}	4.81×10^{-3}	6.29×10^{-4}	6.29×10^{-4}
	σ_R	0	0	21	0
	σ_θ	25802	25865	3390	3371

Fig. 7.1.1 Thick cylinder - Multi-region

A Column Attached to an Elastic Half-Space [33]:

In this problem a long column is attached to an elastic half-space and loaded in tension. Note that the correct solution for the tip deflection of the column is essentially the sum of the extension of the column and the displacement of the half-space under a patch load.

It is of particular interest to note that it was never possible to obtain an acceptable solution to this problem when it was run as a single region, using either linear or quadratic variation. This is due to the fact that, when considered as a single region, much of the column is effectively located at infinity. To obtain accurate results would require a much more extensive mesh on the surface of the half-space, losing all the advantage gained by the use of infinite elements. Assigning the column and the half-space to separate regions eliminates this problem, leading to reasonable results (-5% error) for linear variation and very good results (less than 1% error) for quadratic variation (Figure 7.1.2).



	Displacement at Tip of Beam
Analytical	1.13×10^{-2}
One Region, Linear Variation	5.00×10^{-4}
One Region, Quadratic Variation	6.91×10^{-4}
Two Regions, Linear Variation	1.05×10^{-2}
Two Regions, Quadratic Variation	1.12×10^{-2}

Fig. 7.1.2 A column attached to an elastic half-space

Turbine Component Analysis [34,35]:

In order to evaluate the capabilities of the analysis of real components, an analysis of a commercial cooled turbine blade geometry was carried out.

The blade analyzed is a cooled high turbine blade presently in service. It is subject to mechanical loads (primarily centrifugal) and thermal loads. Of particular interest for this blade is the location and magnitude of the peak stress under the platform. A surface model was built for this problem. Both full and hidden line views of the model are shown in Figure 7.1.3a The model consists of five subregions. The interfaces between subregions are generally perpendicular to the radial direction. The characteristics of the model are summarized below.

Subregion	Elements	Linear (Nodes)	Quadratic (Nodes)
1	60	61	180
2	86	85	256
3	107	98	303
4	106	96	298
5	80	76	232
TOTAL	439	416	1269

The system equations for a fully linear analysis contain 1248 equations.

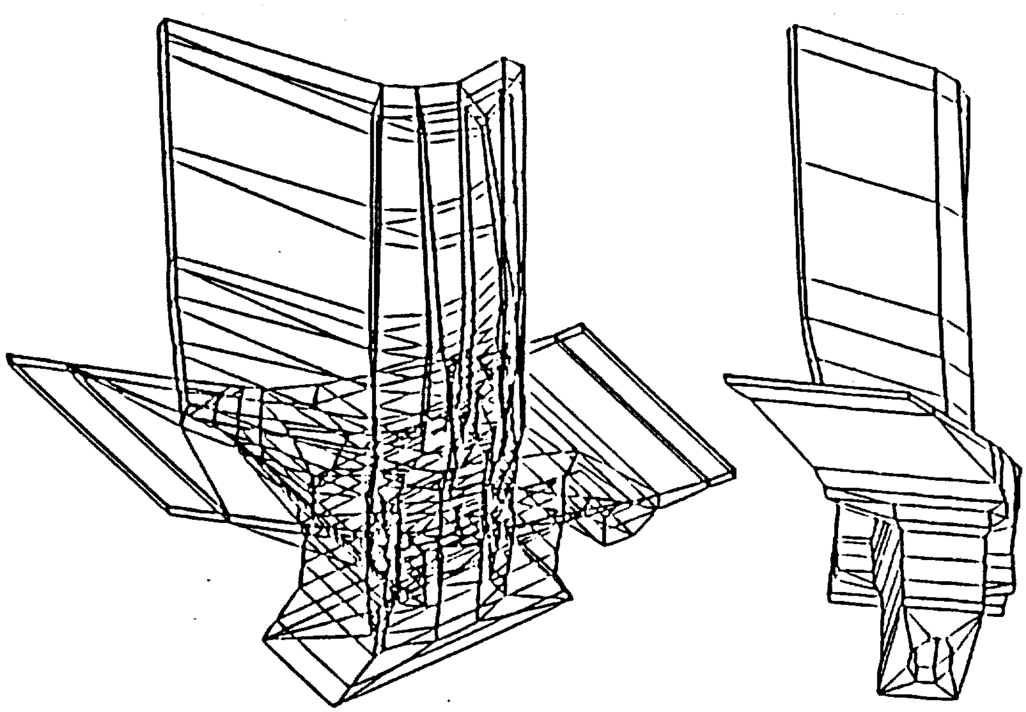


Fig. 7.1.3a BEST3D Model of Cooled Turbine Blade

Initially a fully linear analysis under centrifugal load was carried out. The total centrifugal load at various spanwise stations on the blade was compared with the design calculations for the blade. The agreement between the two totally independent calculations was excellent.

Study of blade tip deflections, load distribution over the base of the blade neck and concentrated stresses indicated reasonable qualitative agreement with three-dimensional finite element results. The time required to execute was only 15 cpu min (on an IBM 3081) compared to 45 cpu min for the finite element (MARC) analysis. While the use of stress contour plots demonstrated that the peak stress in the analysis occurred in the correct location (under the trailing edge rail on the concave side of the blade), the value of the peak stress was too low (146 ksi vs. 169 ksi). Improving this result using a full quadratic analysis would have involved over 3700 degrees of freedom and required 45 to 60 minutes of computing time.

In order to improve local stress accuracy in the critical location while retaining computational efficiency, the mixed variation capability was employed. Quadratic variation was used over sixteen elements (in only two subregions) in the immediate vicinity of the critical rail. The extent of the quadratic variation is shown in Figure 7.1.3b Problem size increased only 6% (from 1248 to 1317 degrees of freedom) and computer time only 10% (to 16.5 cpu min.). The peak stress increased to 174 ksi (on the surface), in excellent agreement with the MARC result of 169 ksi (at a slightly subsurface integration point).

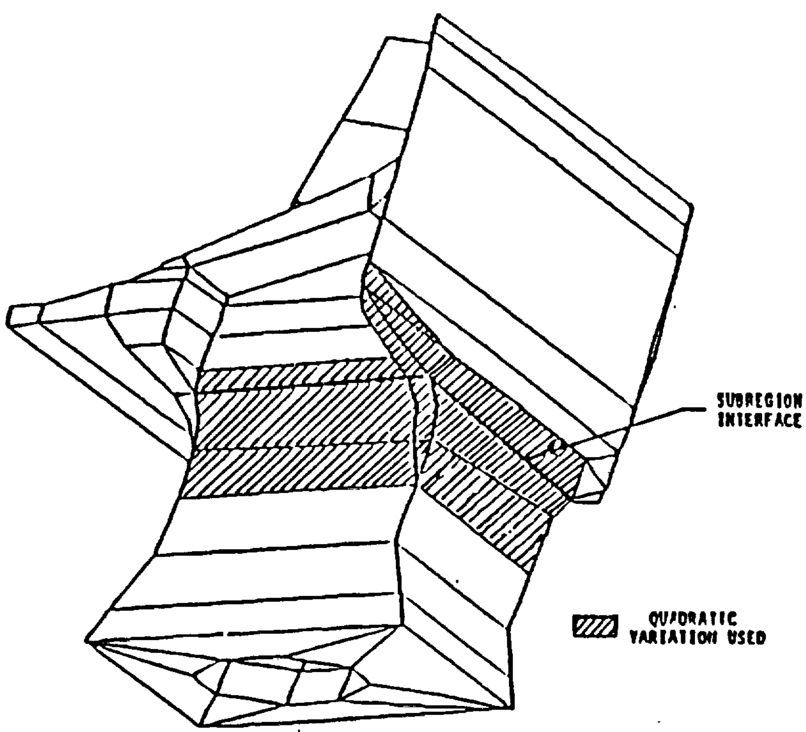


Fig. 7.1.3b Local Quadratic Surface Elements for the **BEST3D** Model of a Cooled Turbine Blade

A cube with a hole [54]:

A test problem of a cube of dimensions 'a' with a cylindrical hole of diameter 'b' was subjected to simple tension such that the deformed shape of the hole is elliptical (Figure 7.1.4a). Figure 7.1.4b shows the discretization used which includes a mixture of 8-noded and 9-noded surface elements. The hole, which is of constant radius, was modeled using the hole element with three nodes in the longitudinal direction. Young's modulus = 100 psi, $\nu = 0.3$ and an applied traction of 100 psi were assumed.

Figure 7.1.4c shows the results of **BEST3D** analysis for various dimensions of the hole compared with **BEST3D** results in which the hole was modeled exactly. The diameter of the hole was varied from b/a = 0.01 to b/a = 0.6. It can be seen that the displacement at the middle node of the loaded face of the cube is in good agreement with the exact 2D modeling. The discrepancy at the larger void ratio is to be expected since the displacement field around the hole is too complex to be modeled by the coarse surface mesh.

Figure 7.1.4d shows the comparison of the displacement variation at the loaded face of the cube. Once again the results for the b/a = 0.1 case are found to be in good agreement with the results of **BEST3D** exact modeling. As expected, results at the middle section of the cube (i.e., nodes 1b, 2b and 3b) are in better agreement with the 2D results.

It was originally thought that the hole element, while perfectly adequate to quantify the loss of stiffness due to their presence, would not be suitable for the determination of the local stress concentration factors. Figure 7.1.4e shows that the actual stress concentration up to a point close to hole surface is quite well predicted and with some additional work, it should be nearly as good as the full 3D idealization.

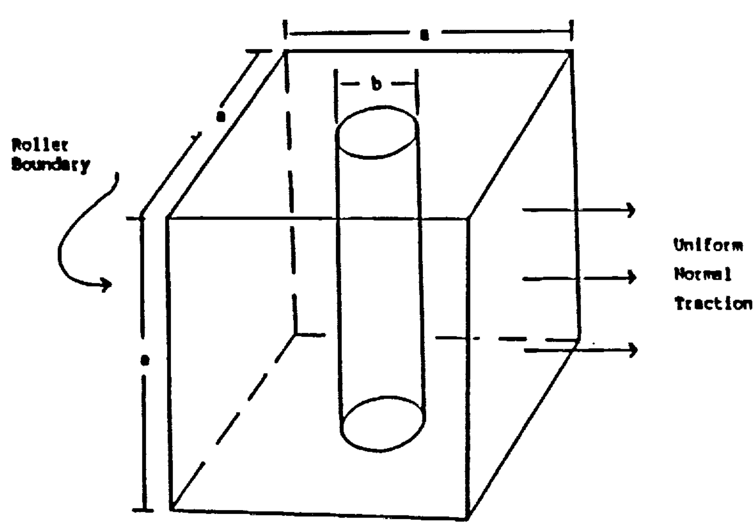


Fig. 7.1.4a Three Dimensional Cube (of length a) with a hole (of diameter b) at the center

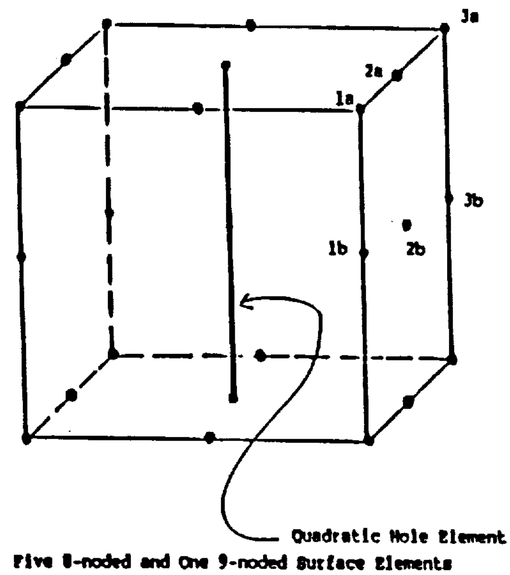


Fig. 7.1.4b Discretization of the cube utilizing quadratic hole element to model the hole

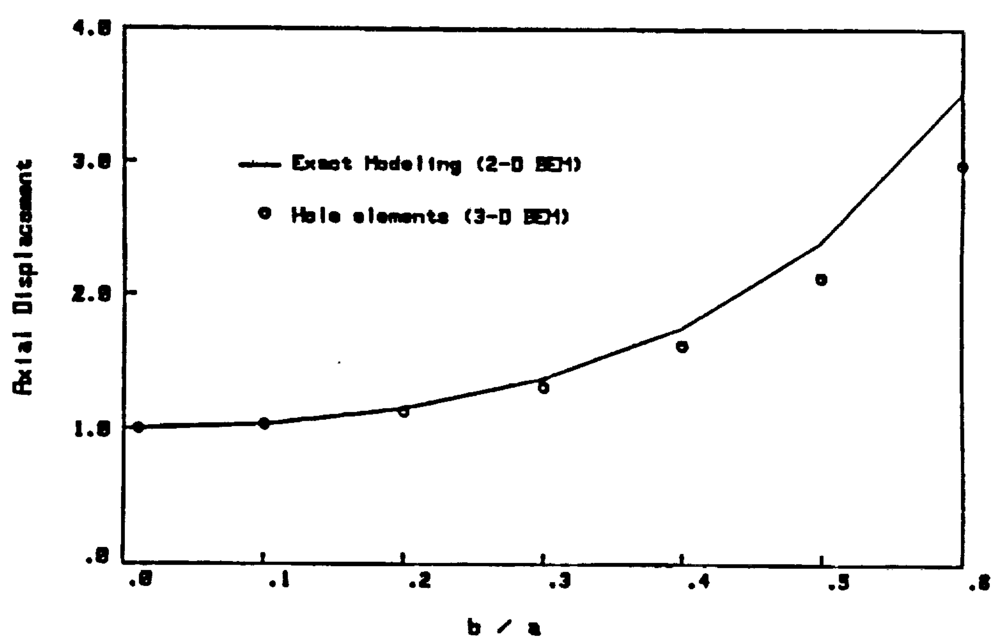


Fig. 7.1.4c Comparison between exact 2-D modelling of the cube and Hole element results for different hole diameters.

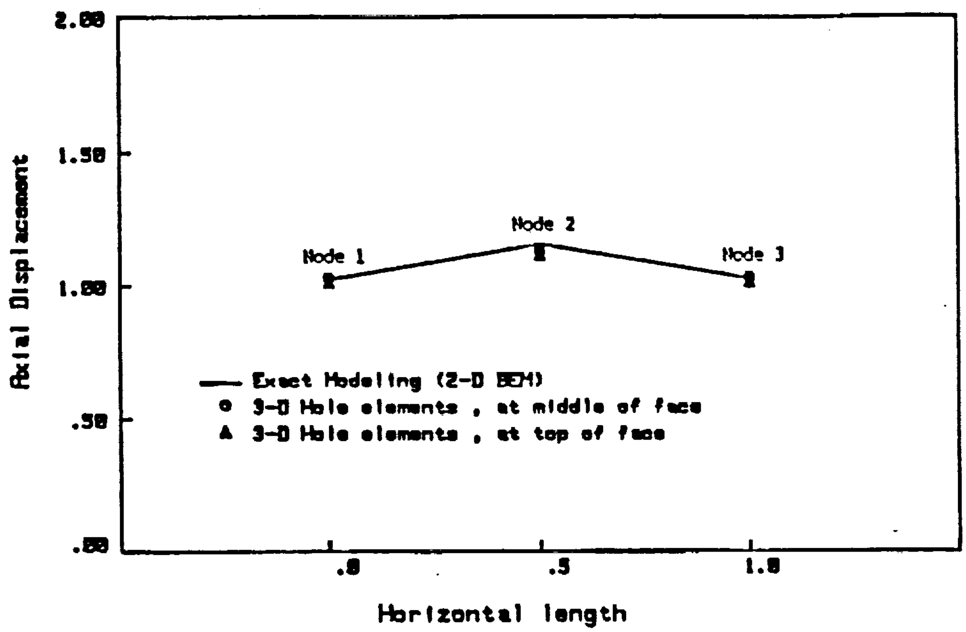


Fig. 7.1.4d Comparison of edge displacements as given by 2-D BEM and the Hole Elements in BEST3D

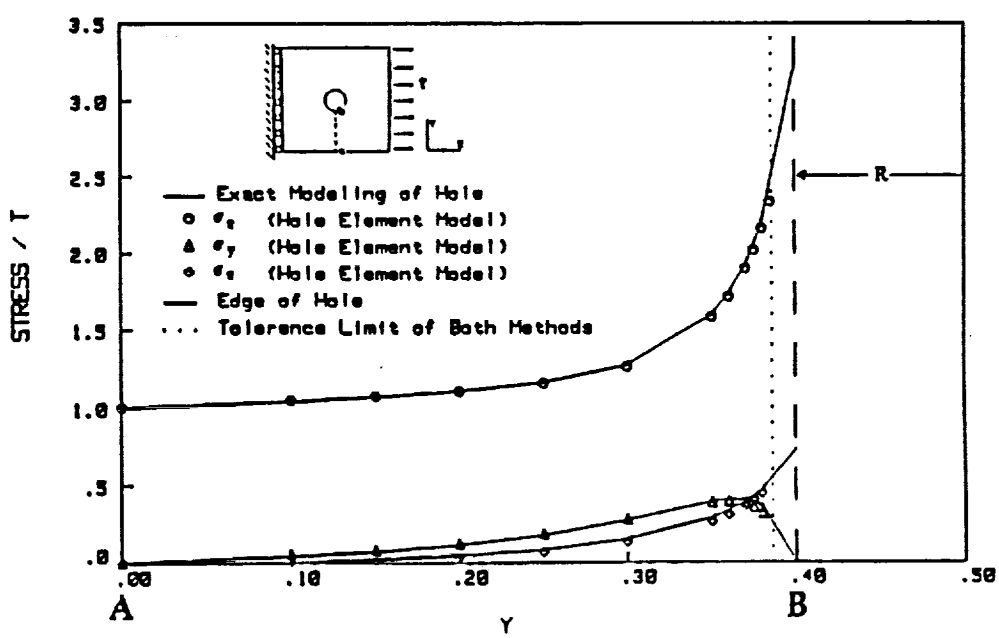


Fig. 7.1.4e Comparison of the Stress Distribution around the Hole between 2-D models and 3-D Hole Element Model

Analysis of a thick cylinder with four holes [54]:

A more complex problem demonstrating the facilities of the current implementation is shown in Figure 7.1.5a, where a 22° segment of a thick cylinder (inner radius = 10 inches, outer radius = 20 inches, height of the cylinder = 2 inches) has four 1 inch diameter holes evenly spaced at radii of $r = 11.25, 13.75, 16.25$ and 18.75. The Young's modulus of the material is $E = 30 \times 10^6$ psi and the Poisson's ratio $\nu = 0.3$. The problem is first modeled with 18 quadratic surface elements and 4 quadratic hole elements. Sixteen (16) of the quadratic surface elements are 8-noded and 2 are 9-noded elements. The cylindrical segment is loaded with an internal pressure of 100,000 psi under the conditions of plane strain, which was simulated by applying rollers on all plane surfaces.

The problem was compared to an axisymmetric, multi-region BEM analysis (using a 2-D code) with a mesh shown in figure 7.1.5b. The radial displacements along the bottom surface is shown in Figure 7.1.5c. Both the axisymmetric and three-dimensional results with holes are in excellent agreement. For the sake of comparison, the results of the solid cylinder are also shown in the graph.

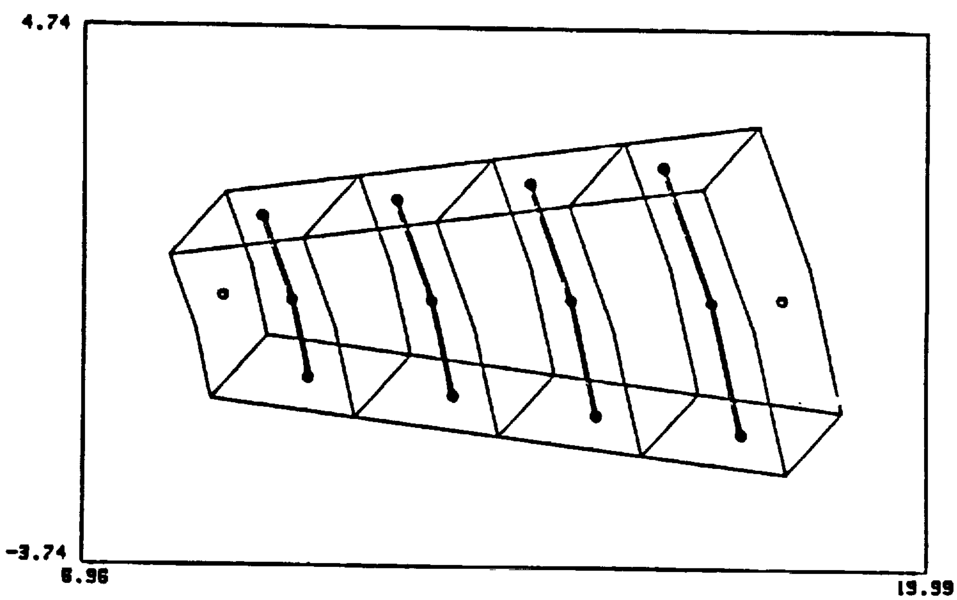


Fig. 7.1.5a Multiple Holes in an Axisymmetric Three-dimensional Block

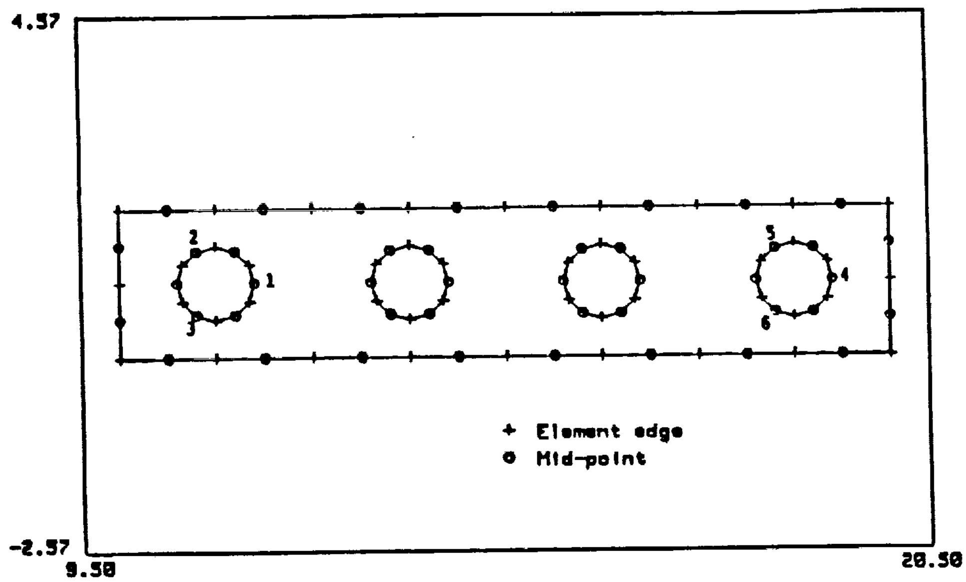


Fig. 7.1.5b Axisymmetric Discretization of the Block with Four Holes

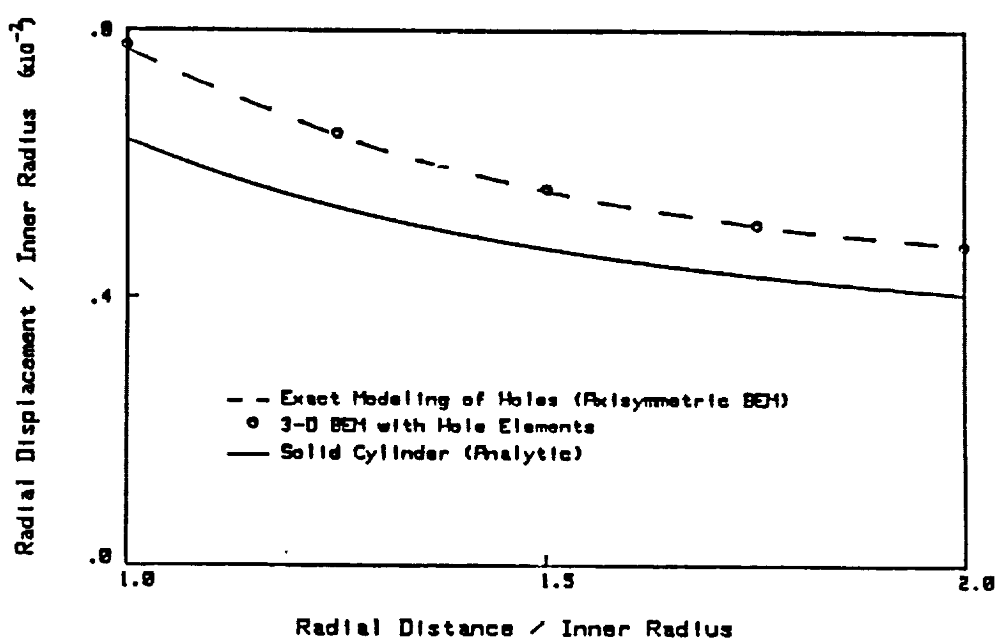


Fig. 7.1.5c Comparison between Axisymmetric results and 3-D modelling with Hole Elements for the 4-hole Block

Cube with a Single Insert [55]:

The first test of the formulation is on a unit cube with a single insert through its center of radius 0.1. The cube is subjected to tension and shear in the direction parallel and perpendicular to the insert. The cube has a modulus of 100.0 and a Poisson ratio of 0.3. Consistent units are used for all information described in this problem. The Poisson ratio of the insert is assumed to be the same as that of the cube.

The problem is analyzed by both the present formulation and by a full three- dimensional multi-region BEM approach. As shown in Fig. 7.1.6a, the model for the insert formulation consists of fourteen quadratic boundary elements and the insert contains three quadratic insert elements. The two-region, three-dimensional model shown in Fig. 7.1.6b contains twenty quadratic boundary elements in the first region and sixteen in the second. Note 9-noded elements are used in describing the insert and hole to accurately capture the curvilinear geometry.

In Fig. 7.1.6c, the profile of the end displacement of the cube under a uniform normal traction of 100.0 (in parallel with the insert) is shown. The present formulation is in good agreement with the full three-dimensional results for Ei/E = 10. For the case Ei/E = 100, the insert formulation exhibits greater stiffness than the 3D results. This difference is contributed by the way the load is distributed from the insert to the composite matrix. In the full 3D model, the applied traction and the resulting reactions at the fixed end act directly on the end of the insert. In the composite formulation, the insert is assumed not to intersect the body surface and therefore the insert is moved back slightly from the end of the cube. The load is therefore transferred through the composite matrix to the end of the insert and to its sides in a manner that is slightly different from the full 3D analysis.

In Fig. 7.1.6d, the stress distribution through the center of the cube (from A to B as indicated in the figure) is shown. Again the results are very good for Ei/E = 10, and deviates slightly from the full 3D results in the second case.

In Figs. 7.1.6e and 7.1.6f, the lateral displacements along the side of the cube are shown for a cube subjected to shear traction of 100. For the case of applied shear perpendicular to the insert (Fig. 7.1.6e), the results for both the insert and full 3D model show good agreement. Once again a slight deviation is observed for Ei/E = 100. In the case of the shear traction in the plane of the insert (Fig. 7.1.6f) the insert has little effect on the displacement (as anticipated) and all results fall in close proximity.

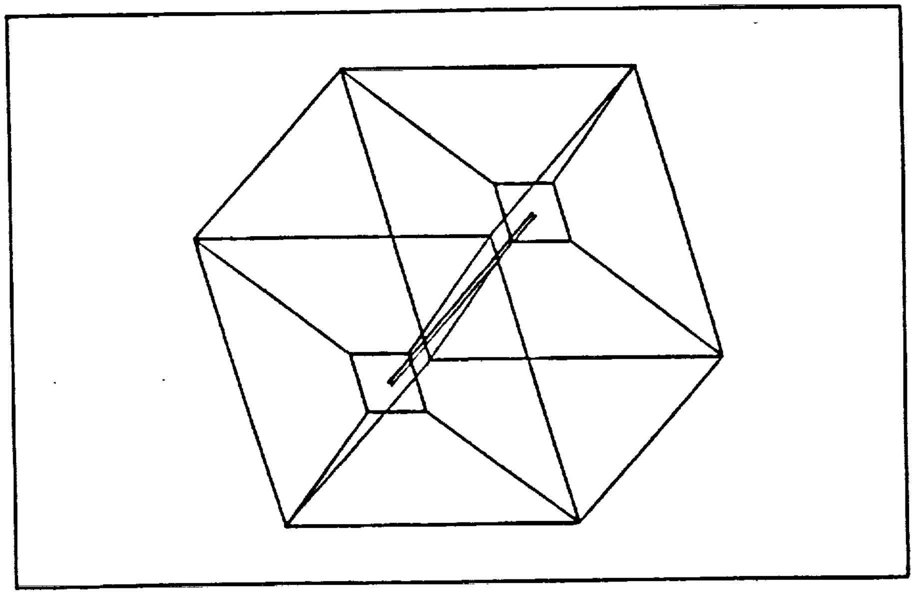


Fig. 7.1.6a Discretization of an Insert in a Unit Cube Utilizing Quadratic Insert Elements

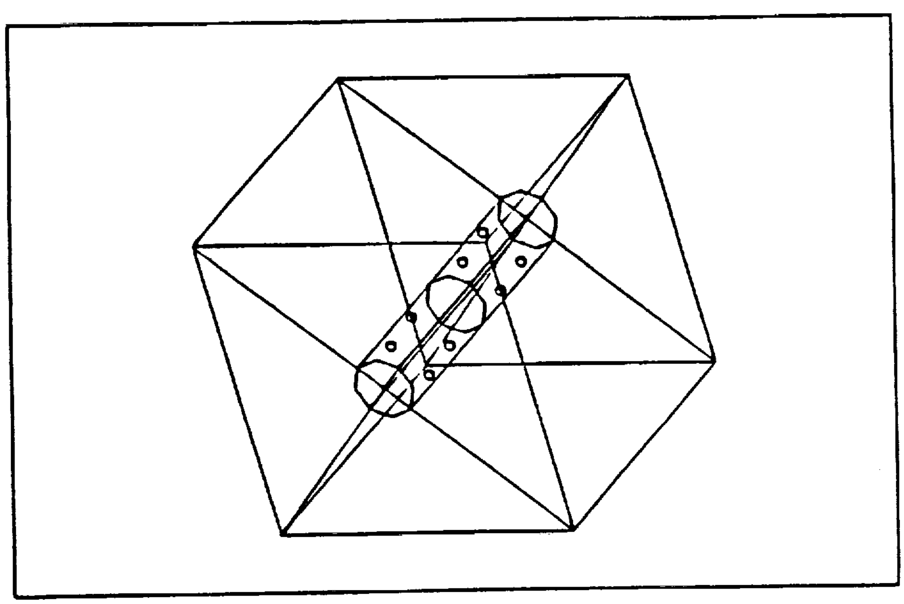


Fig. 7.1.6b Full Three-Dimensional Multi-region Discretization of an Insert in a Unit Cube

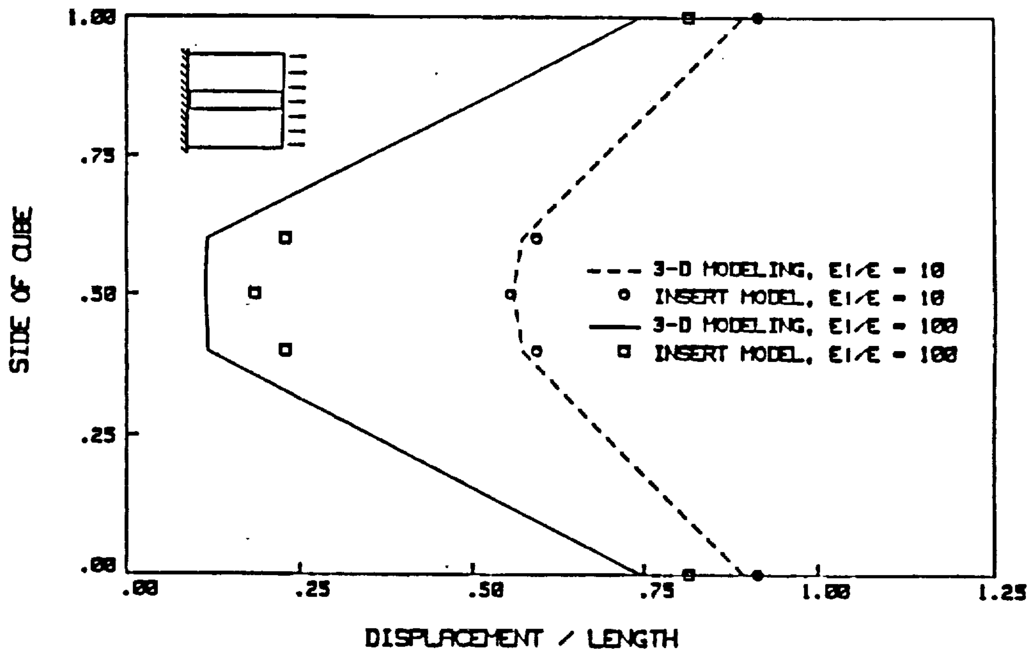


Fig. 7.1.6c Comparison of Displacement Profiles between the full 3-D model and the Insert Element Model for the End of a Cube in Tension

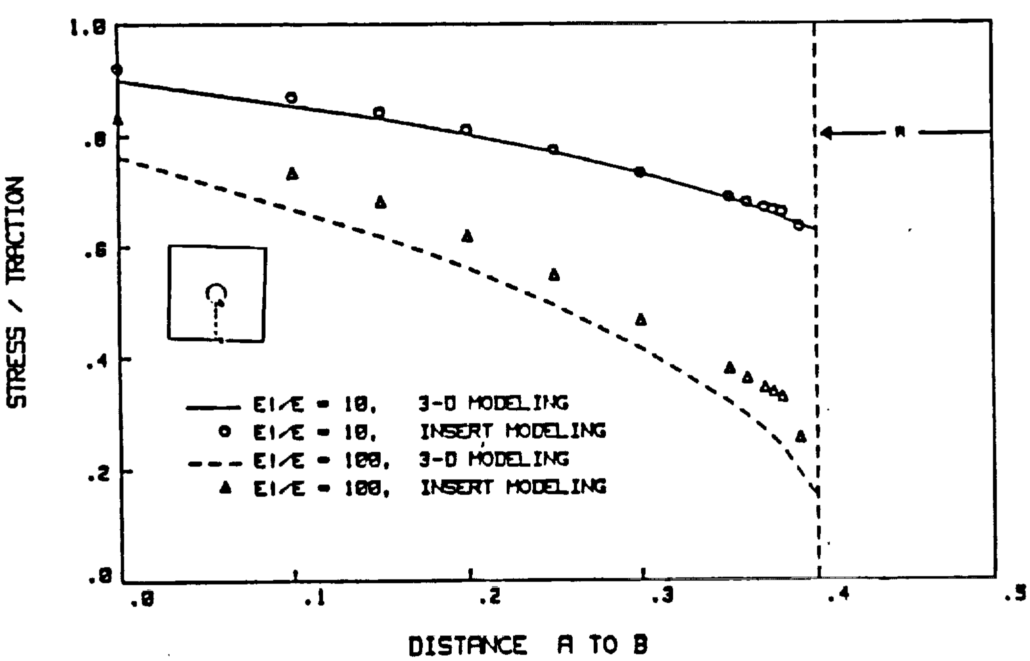


Fig. 7.1.6d Axial Stress through the Cross Section of a Unit Cube in Tension with a Single Insert in Parallel with the Loading

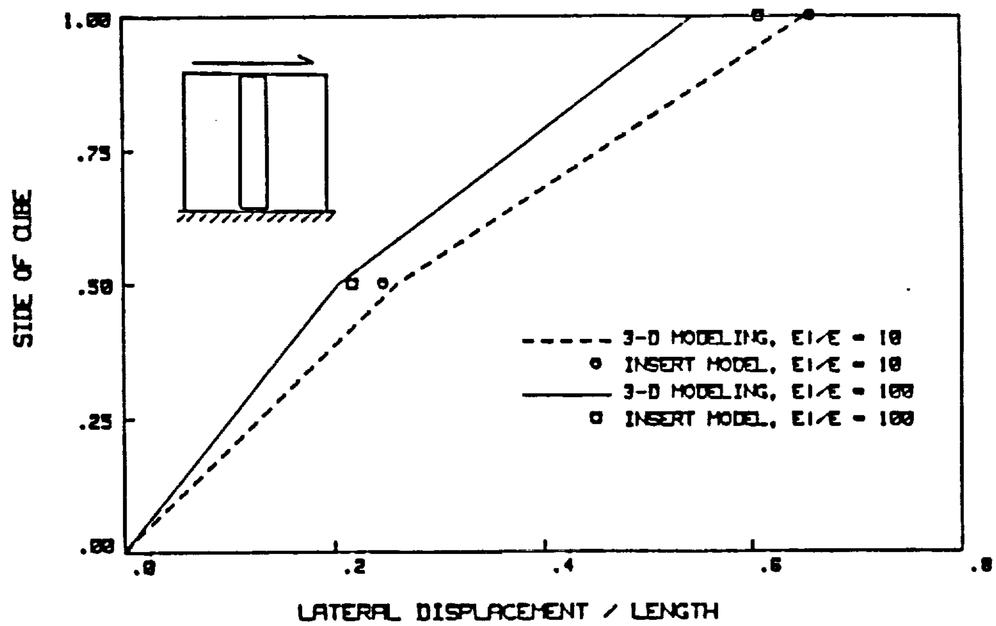


Fig. 7.1.6e Lateral Displacement along a side of a Cube subjected to a Shear Force Perpendicular to an Insert

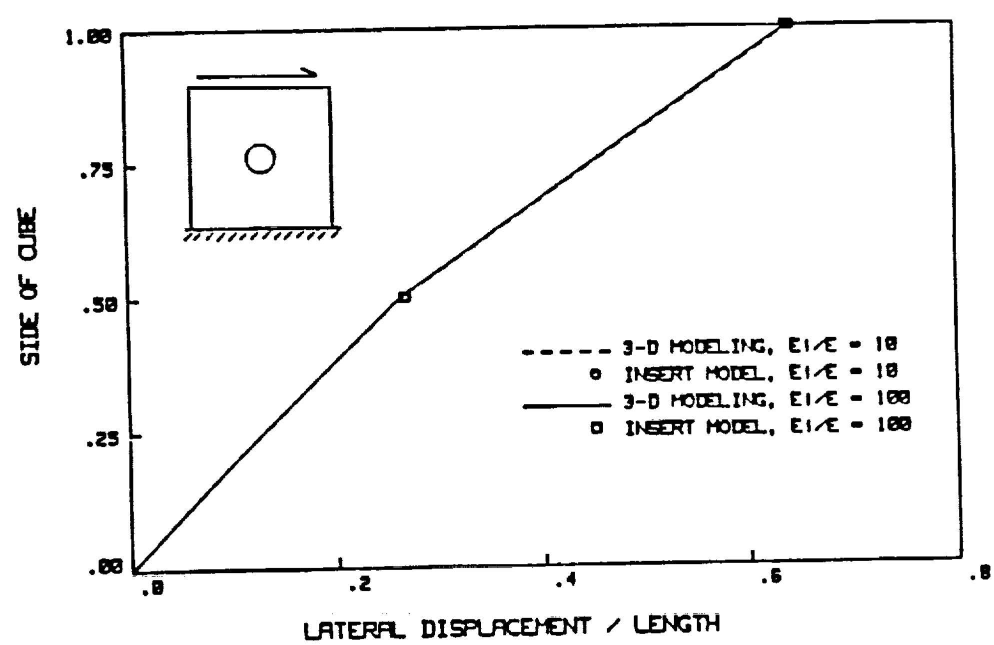


Fig. 7.1.6f Lateral Displacement along a side of a Cube subjected to a Shear force in the Cross-Plane of an Insert

Lateral Behavior of a Cube with Multiple Inserts [55]:

Existing methods of analysis of composite material based on mechanics of materials have been relatively successful in predicting the behavior of composite material for loading in the longitudinal direction. The properties perpendicular to the direction of the fibers are not so readily predictable by present means. The focus of the present example concerns this lateral behavior.

Four cubes (Fig. 7.1.7a) with one, two, five and nine inserts are fixed with a roller boundary condition on one side and subjected to a uniform traction, perpendicular to the inserts. The material properties, given in consistent units, are

$$E^{insert} = 10000. \quad E^{matrix} = 100.$$

$$\nu^{insert} = 0.3 \qquad \nu^{matrix} = 0.3$$

For the cube with one and two inserts, the boundary mesh consists of two quadratic surface elements on each lateral side and four elements on the top and bottom. For the cubes with five and nine inserts, one additional element was added to the side with the roller boundary condition. The top and bottom faces contain six elements to match the pattern of the sides. In all cases, each insert contained three one-dimensional quadratic elements.

The profile for the end displacement for a cube with one insert and five inserts are shown in Figs. 7.1.7b and 7.1.7c The results seem to be in good agreement with the two-dimensional results. The 2D results are approximations since plane stress is assumed. The 3D solutions for the one insert are within 2% error of the 2D solution and 6% for the case of nine inserts where the insert of volume to total volume ration is 28.2%. The result is also displayed in a plot of Effective Modulus vs. Insert Volume Ratio in Fig. 7.1.7d The effective modulus is defined as the average stress/average strain. The three-dimensional results follow closely to the two-dimensional solution.

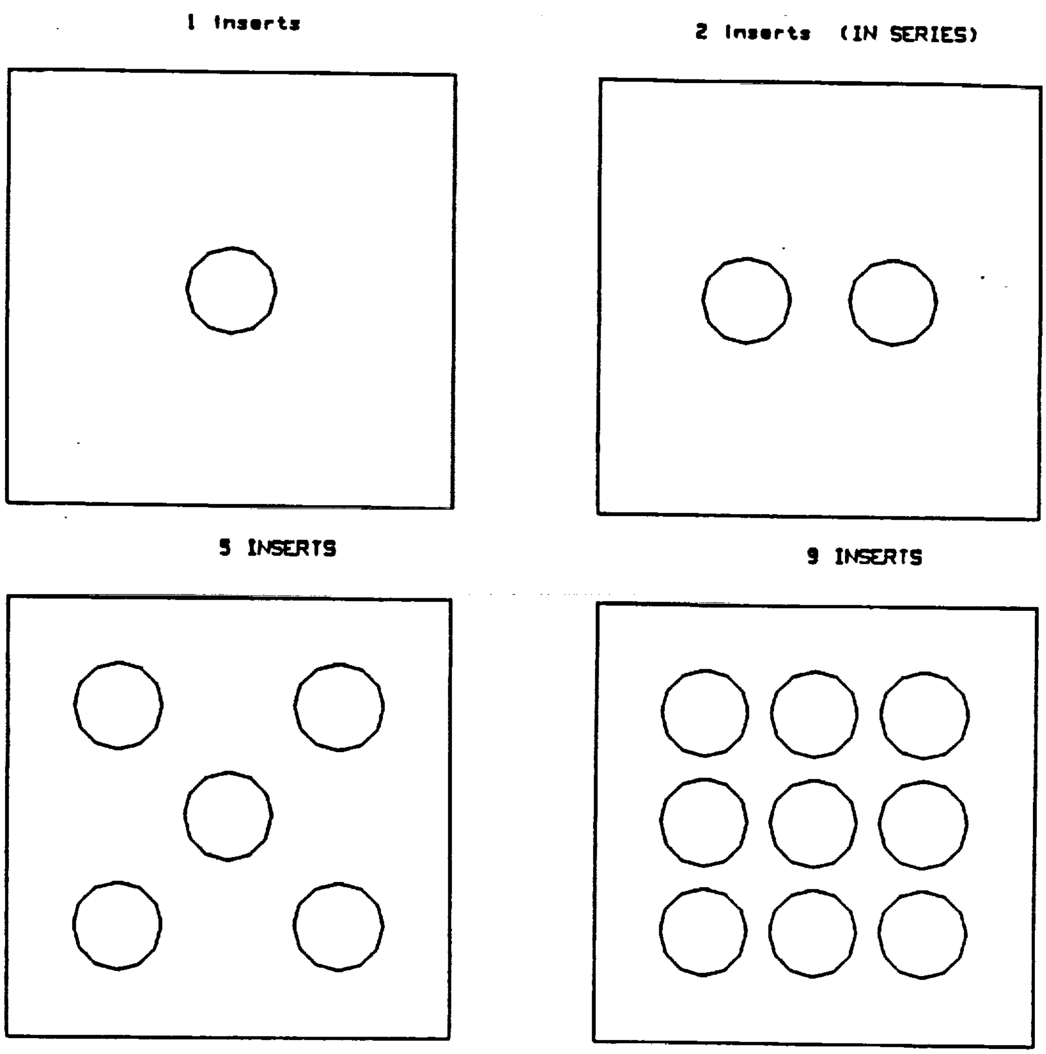


Fig. 7.1.7a Arrangement of Multiple Inserts in a Unit Cube Subjected to Lateral Tension

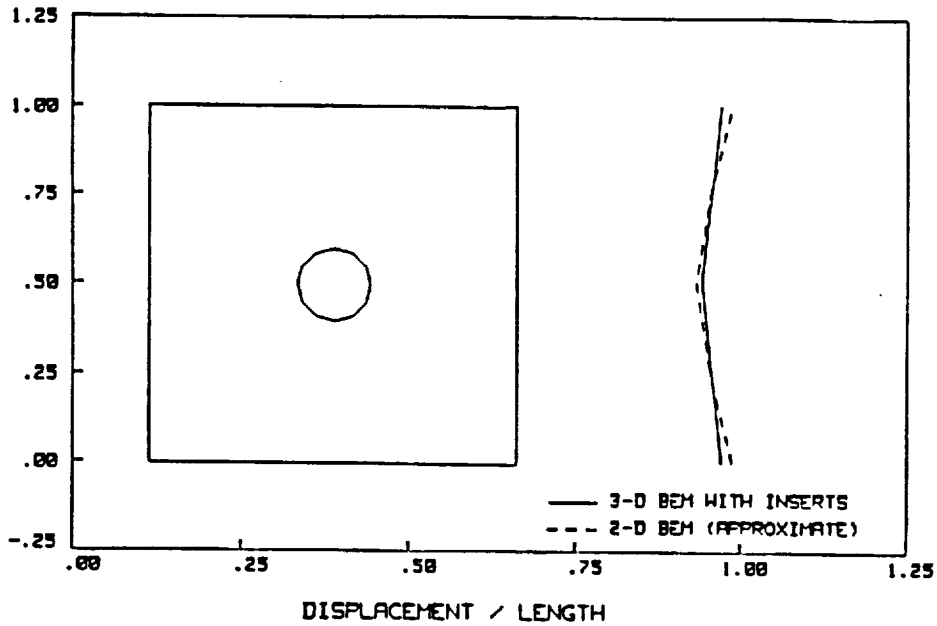


Fig. 7.1.7b Displacement Profile of a Cube with a Single Insert under Lateral Tension

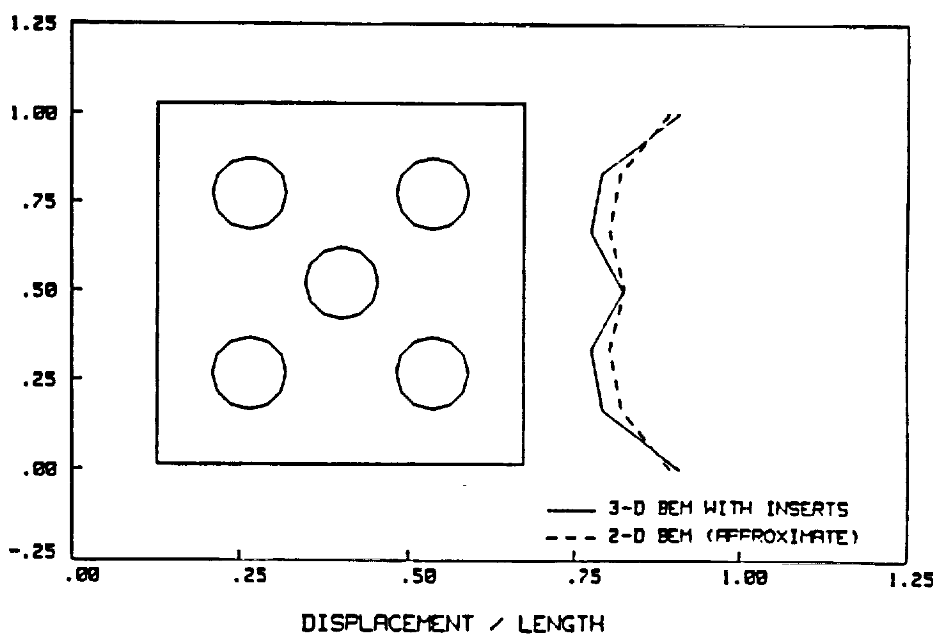


Fig. 7.1.7c Displacement Profile of a Cube with Five Inserts under Lateral Tension

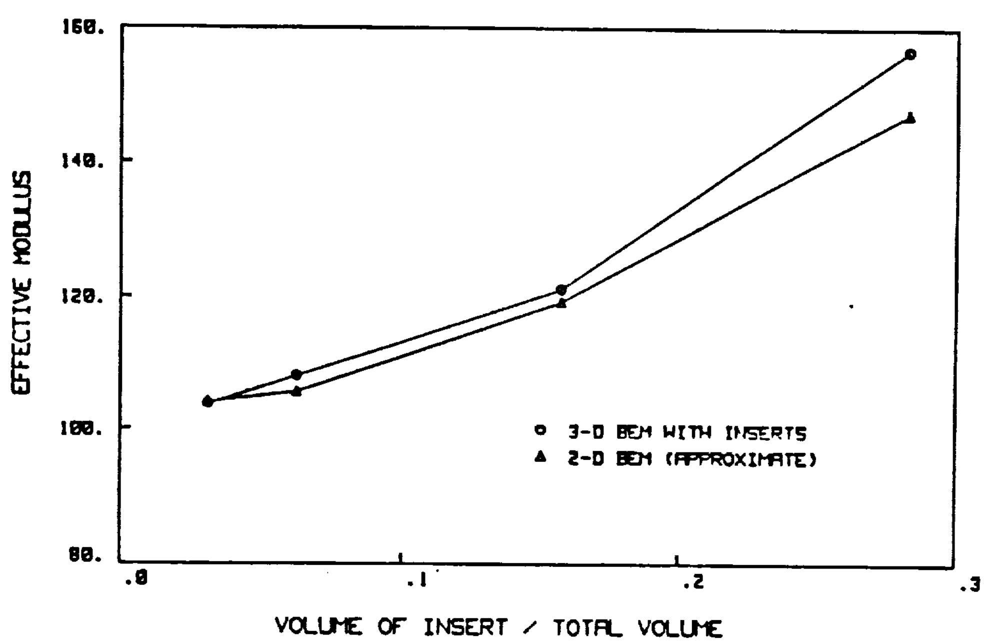


Fig. 7.1.7d Effective Transverse Modulus of a Cube as a Function of Insert Volume to Total Volume

Thick Cylinder with Circumferential Insert Supports [55]:

The strength of a cylinder under internal pressure can be increased by adding stiff circumferential insert supports. In the present example, a three-dimensional, open ended thick cylinder with four inserts is analyzed. The inner and outer radii of the cylinder are 10 and 20 respectively, the thickness is 2 and the radius of the fully-bonded inserts is 0.5. By using roller boundary conditions on the faces of symmetry, only a fifteen degree slice of the thick cylinder needs to be modeled. As shown in Fig. 7.1.8a, sixteen eight-noded quadratic boundary elements are used to define the sides of the model, a nine-noded element is used on both the internal and external faces of the cylinder, and three insert elements are used per insert. Note, the inserts in this problem are curvilinear in geometry. The elastic modulus of the cylinder is assumed to be 100, and the effect of inserts with five different moduli of 100, 250, 500, 750 and 1000 is studied. The Poisson ratio is 0.3 for both the composite matrix and insert, and the internal pressure in the cylinder is 100.

Results from a multi-region, axisymmetric BEM analysis (using a 2-D code) were used for comparison with the 3D insert results of the present example. The axisymmetric model consists of twenty quadratic boundary elements on the outer surface, and six boundary elements per hole and per insert (Fig. 7.1.8b). The radial displacement through the thick cylinder along the top face is shown in fig. 7.1.8c for all five moduli. The displacement for the composites with low Ei/E ratios are in good agreement with the axisymmetric results, and diverge slightly for higher Ei/E ratios. In Fig. 7.1.8d, the circumferential stress is shown for the same points along the top edge. This stress is smooth for the homogeneous case (Ei/E = 1.0) and exhibits increasing fluctuations as the Ei/E ratio increases and the inserts take on more of the load. The circumferential stress of the 3D insert model is in good agreement with the axisymmetric results for all cases. In Fig. 7.1.8e, the radial stress is displayed for the two models. The inserts have little effect on this stress and the curves for the five moduli fall close together for both approaches.

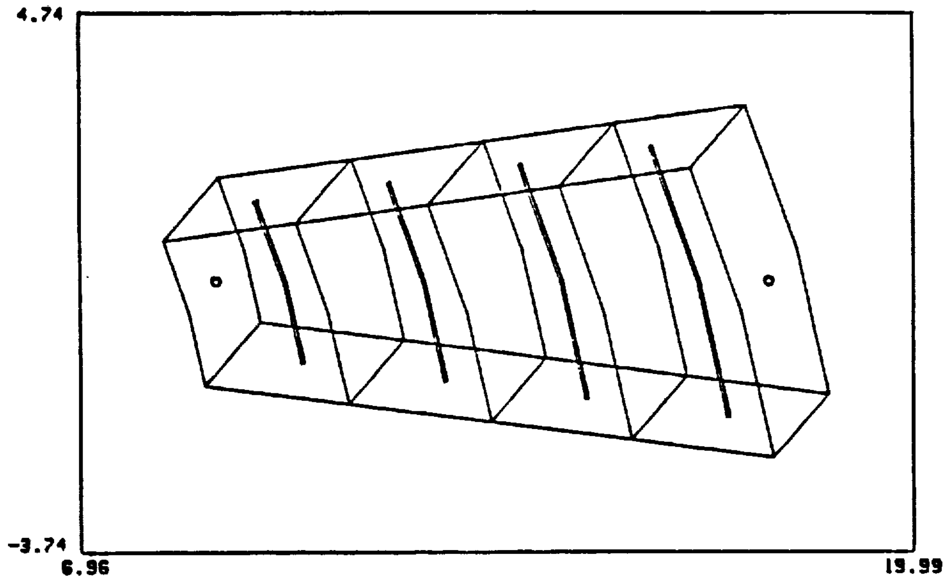


Fig. 7.1.8a Discretization of a Thick Cylinder with Four Inserts Utilizing Insert Elements

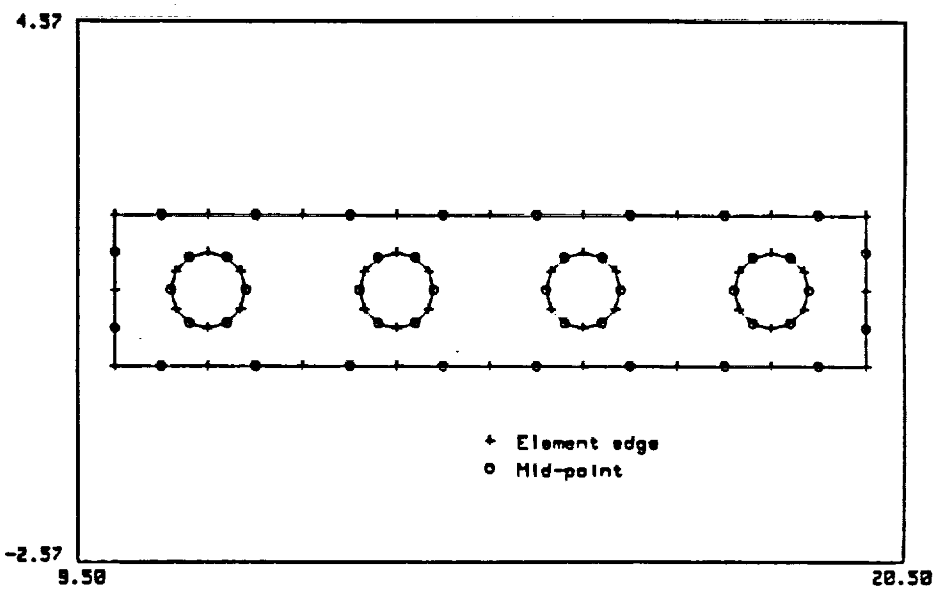


Fig. 7.1.8b Axisymmetric Multi-region, Discretization of a Thick Cylinder with Four Inserts

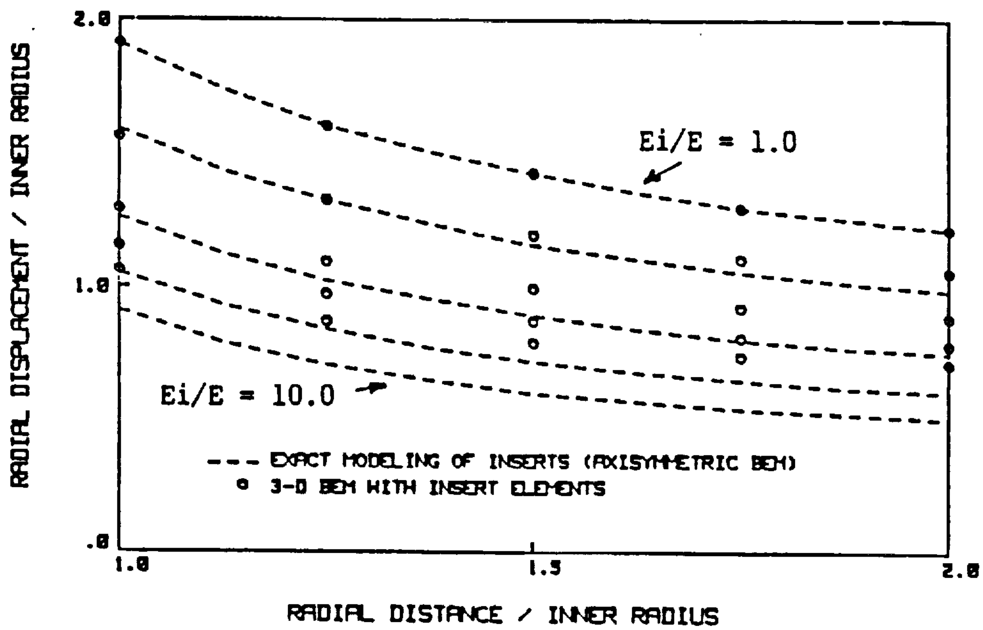


Fig. 7.1.8c Radial Displacement Through a Pressurized Thick Cylinder with Circumferential Inserts for $E_i/E = 1.0,\ 2.5,\ 5.0,\ 7.5,\ 10.0$

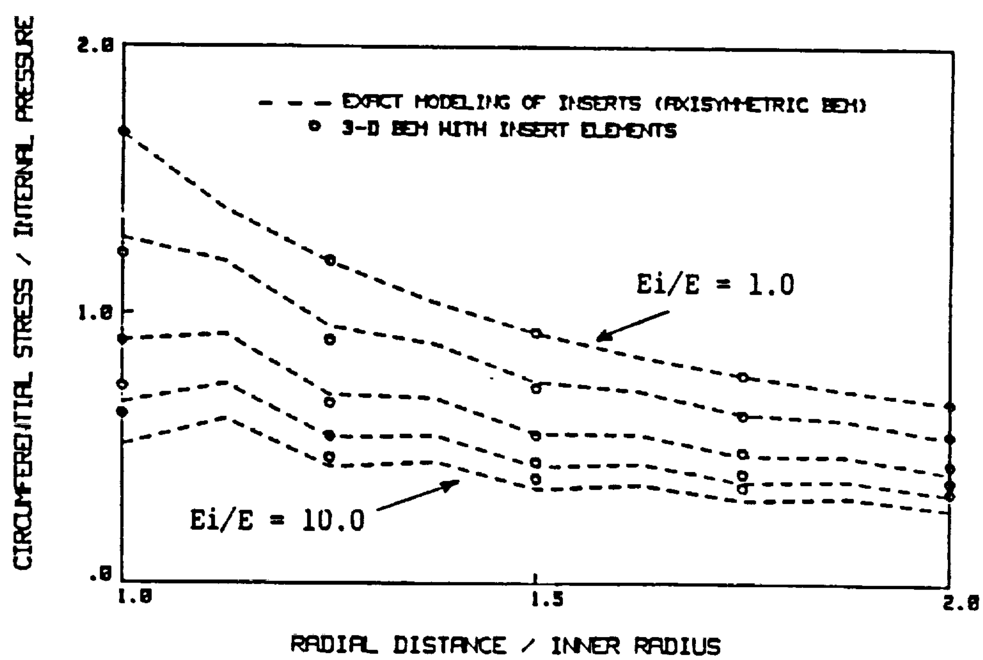


Fig. 7.1.8d Circumferential Stress Through a Pressurized Thick Cylinder with Circumferential Inserts for $E_i/E = 1.0,\ 2.5,\ 5.0,\ 7.5,\ 10.0$

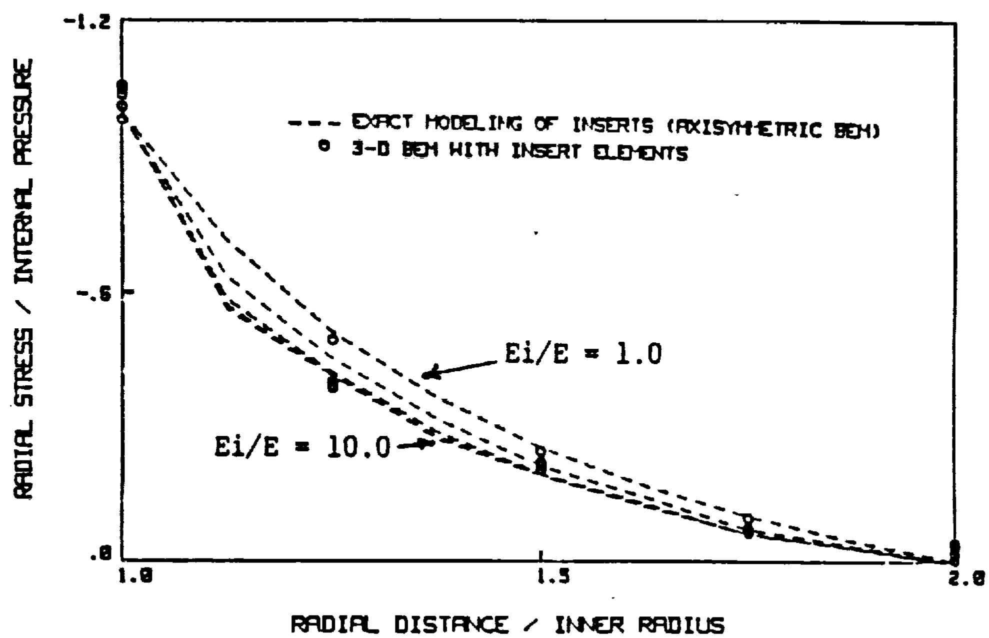


Fig. 7.1.8e Radial Stress through a Pressurized Thick Cylinder with Circumferential Inserts for $E_i/E = 1.0,\ 2.5,\ 5.0,\ 7.5,\ 10.0$

Cube with Multiple Inserts with Random Orientation [55]:

In an attempt to analyze a material with a random fiber structure, cubes with multiple inserts oriented in random directions are studied. The cubes are of unit length and have four boundary elements per side (Fig. 7.1.9a). Randomly oriented fibers of variable length with radii of 0.05 are placed in five cubes in quantities of 5, 10, 15, 20 and 25 (Fig. 7.1.9b-f). Three cases of material properties are considered for each cube. The modulus of the composite matrix is 100 for all cases, however, the modulus of the inserts are 500, 10,000 and 200,000 in the three cases studied. Poisson's ratio is uniformly 0.3 throughout. Roller boundary conditions are employed on three adjacent sides and a uniform normal traction of 100 is applied to a fourth face.

The normal end displacement at the center of the face on the side with the applied traction is plotted against the number of inserts in a cube for the three materials (Fig. 7.1.9g). The displacement decreases with increasing number of inserts per cube and increasing E_i/E values as expected.

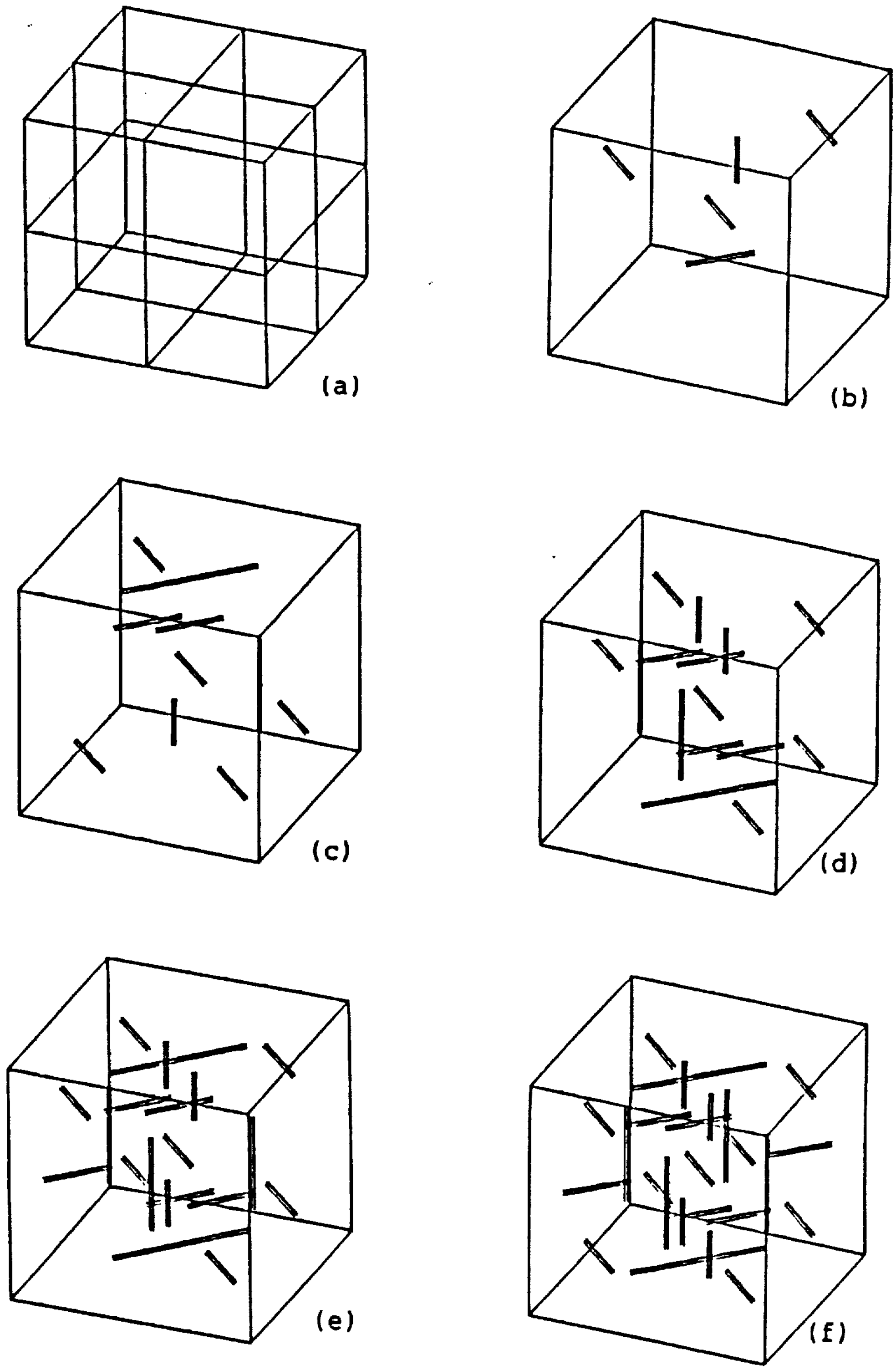


Fig. 7.1.9 (a) Surface Discretization of a Unit Cube Used in the Study of Random Oriented Inserts,

(b,f) Orientation of Variable Lenght Inserts within Unit Cubes containing 5, 10, 15, 20, and 25 Inserts, respectively

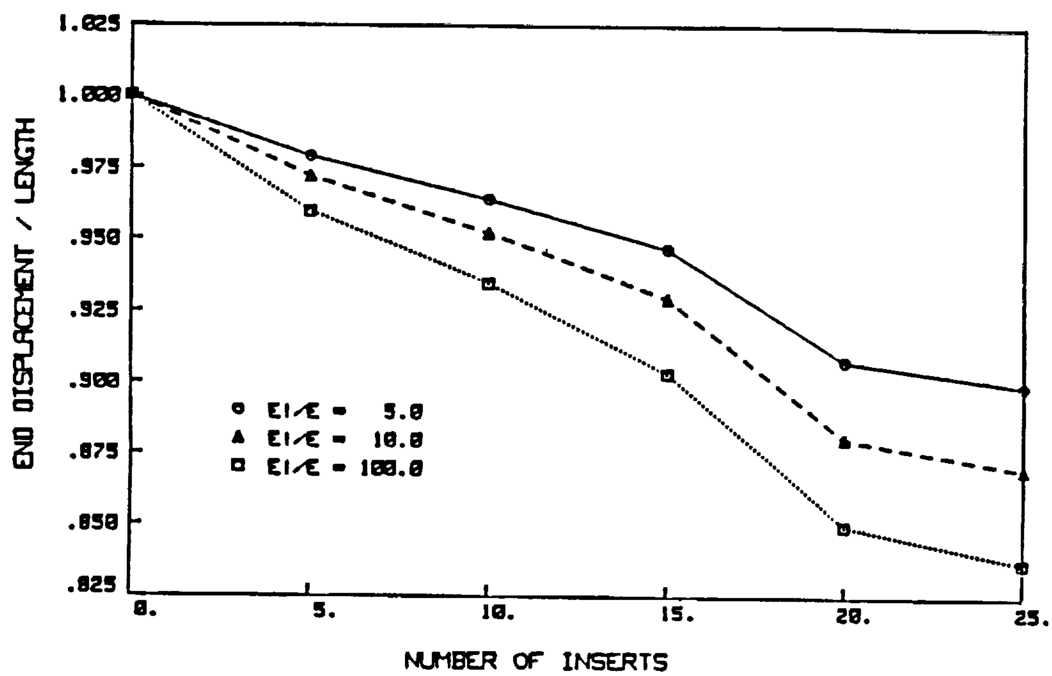


Fig. 7.1.9g End Displacement of a Unit Cube Used in the Study of Random Oriented Inserts of $E_I/E = 5, 10, 15, 20$, and 25

Beam with Insert Reinforcement in Bending [55]:

In the last example, the applicability of the present formulation to the study of the micromechanical behavior of the ceramic composite is apparent. the present formulation, however, is equally applicable to typical problems encountered by civil engineers. Using Composite-BEST reinforced concrete can now be modeled exactly as a three-dimensional body and studied in detail for the first time. The present example considers a reinforced concrete beam. Here the concrete plays the role of the composite matrix and the reinforcement bars play the role of the fiber insert. In Fig. 7.1.10a, a 4x1x1 beam with four inserts is modeled using twenty-eight quadratic boundary elements. The ratio of insert modulus to matrix modulus (Ei/E) is studied for a range of valves between 1 and 100. The Poisson ratio is 0.3 for both the beam and reinforced rods.

The beam is completely fixed at one end and a downward shear traction of 100 is applied to the other end. The non-dimensional vertical displacement of the end obtained from the analysis is shown in Fig. 7.1.10b as a function of E_i/E . The non-dimensional displacement is defined here as the end displacement of the reinforced beam divided by the displacement of a homogeneous beam under similar conditions.

The end displacement obtained from the mechanics of material solution is also displayed in Fig. 7.1.10b in non-dimensional form. The curvature of the two plots are very similar but differ in magnitude. This difference is contributed to the fact that although the mechanics of material solution accounts for the stiffening due to the inserts, it does not include the effect of interaction between inserts.

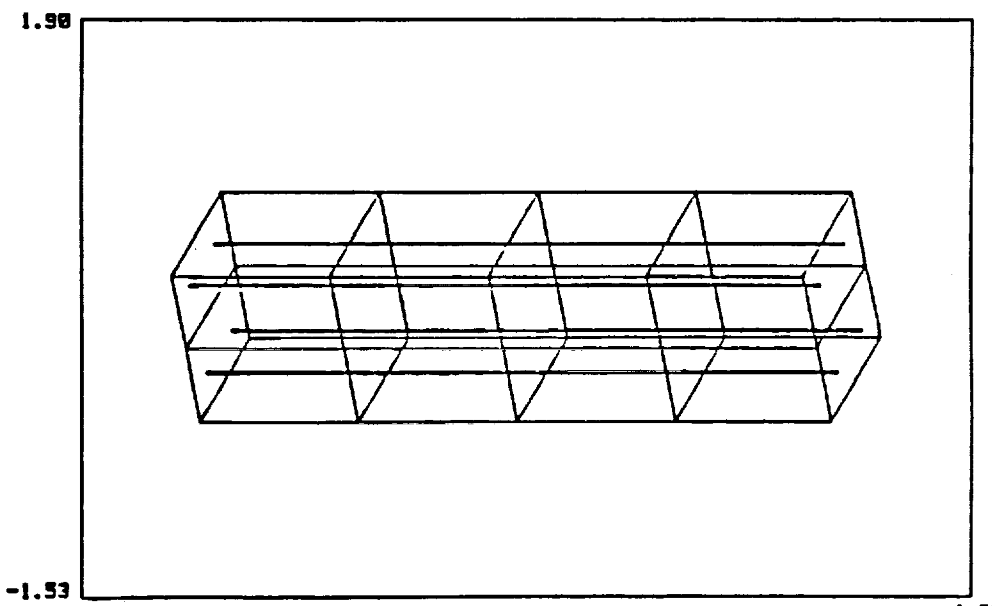


Fig. 7.1.10a Discretization of a Reinforced Beam Utilizing Quadratic Insert Elements to Model the Four Inserts

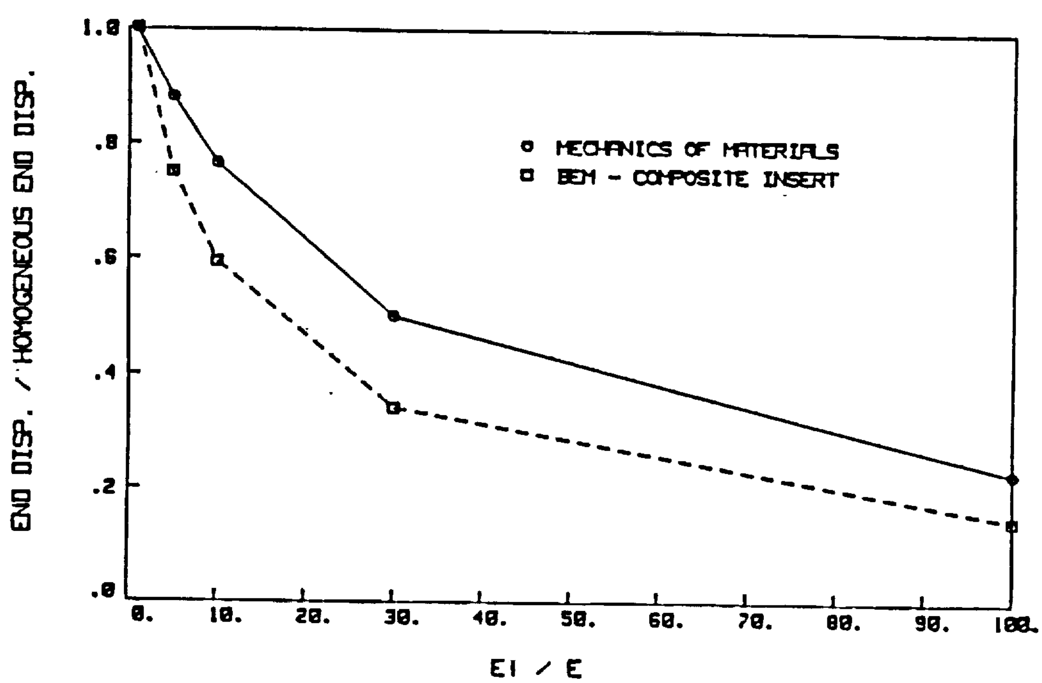


Fig. 7.1.10b Non-dimensionalized Vertical End Displacement of a Reinforced Beam in Bending vs. the Modulus of the Insert over the Modulus of the Beam

Laminated Fiber Composite [55]:

A laminated composite fabricated from a fiber composite material is shown in Fig. 7.1.11a. The fiber composite is constructed with a single row of fully-bonded fibers oriented in the same direction. A two-ply laminate is then constructed from the fiber composite with the fibers of the two layers oriented at 90° angles. A boundary element model created for the study of this material is shown in Fig. 7.1.11b. A small slice containing two inserts in each layer is used. The model consists of two regions. The outer surface of each region is modeled with sixteen quadratic boundary elements and each insert contains two quadratic insert elements. The interface between the two regions is assumed to be a perfect bond, however, the present version of the program allows for sliding and spring connections also.

The Composite structure is subjected to bi-axial tension. This is accomplished with normal tractions of 100 applied to two adjacent roller boundary conditions applied to the opposite ends. The elastic modulus of the composite matrix of both regions are assumed to be 100, and the moduli of the inserts vary between 100 and 10,000. The Poisson ratio is 0.3 for both the composite matrix and inserts at all times.

Figure 7.1.11c displays the displacement as a function of insert moduli for a point on the interface at the corner of the plate adjacent to the sides with the applied traction. The material exhibits less displacement as the modulus is increased, as expected.

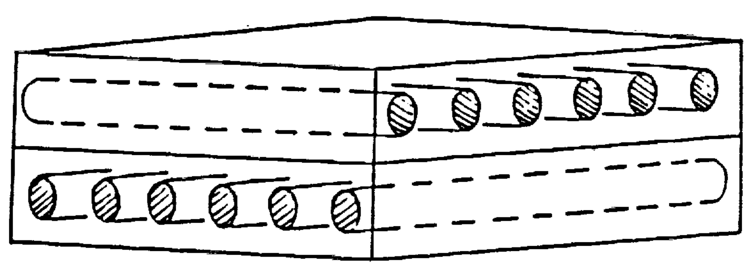


Fig. 7.1.11a Laminate-Fiber Composite

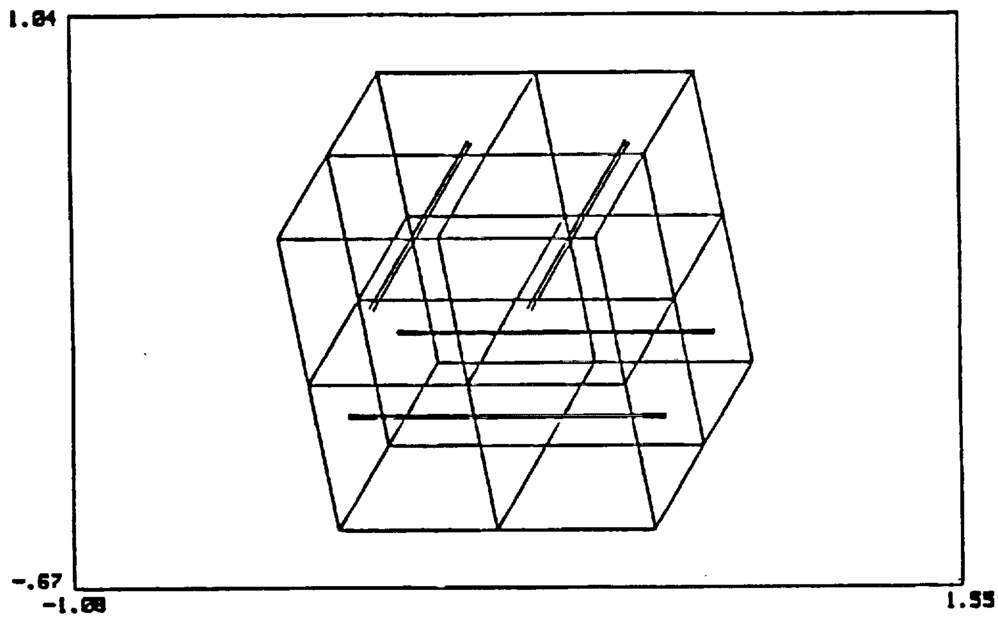


Fig. 7.1.11b BEM Discretization of a Laminate-Fiber Composite

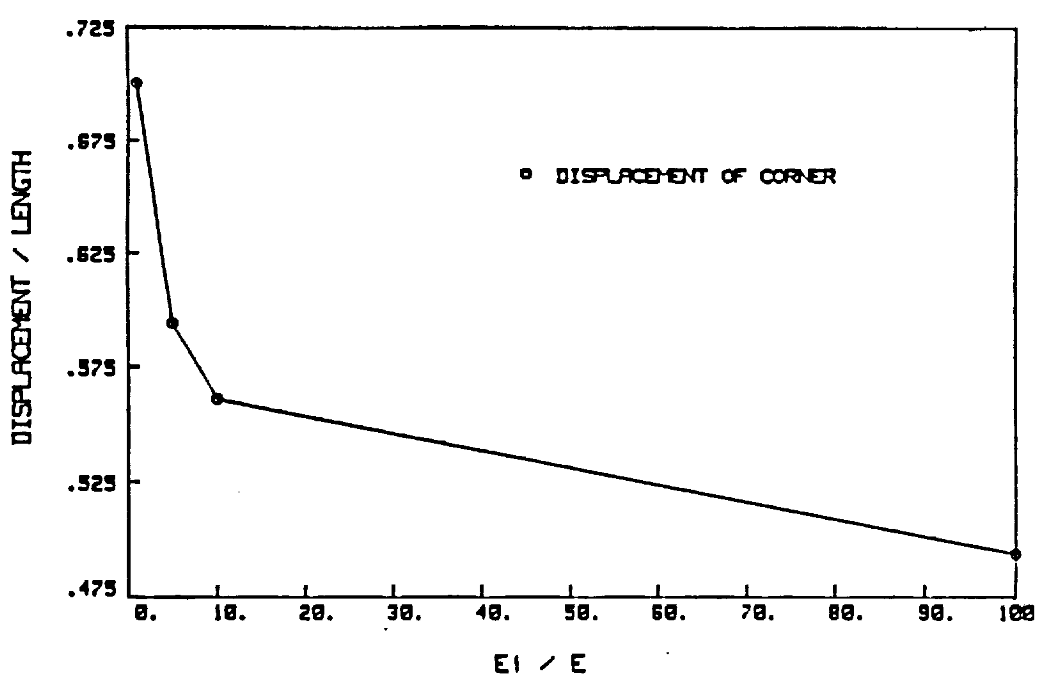


Fig. 7.1.11c Lateral Displacement of a Point at the corner of the interface of a Laminate-Fiber Composite under Bi-Axial Tension

7.2	VIBRATION ANALYSIS

Free-vibration of a rectangular parallelepiped [53]:

Mesh sensitivity studies for the **BEST3D** natural frequencies and mode shapes capability were done using three-dimensional solutions for cantilevered rectangular parallelepipeds obtained by Leissa and Zhang [56] using a Ritz technique. In their paper they report the first five modes of each type (torsion, easy bending, stiff bending and extension) for each of five geometries. Convergence studies indicate that their results are accurate to better than 1% for the first mode.

One of the cases studied is a parallelepiped (Fig. 7.2.1) of dimensions 1 × 1 × 0.5. One end of the structure is completely fixed in all three directions. Four single region, quadratic **BEST3D** models, summarized in Table 7.2.1a were used in the study. The results are tabulated (Table 7.2.1b) with mode type and variation from the Ritz solution listed. It can be seen that rapid convergence is achieved for the first mode of each type by the addition of elements in the axial direction. Convergence of higher modes requires the use of additional elements on the lateral surface of the parallelepiped.

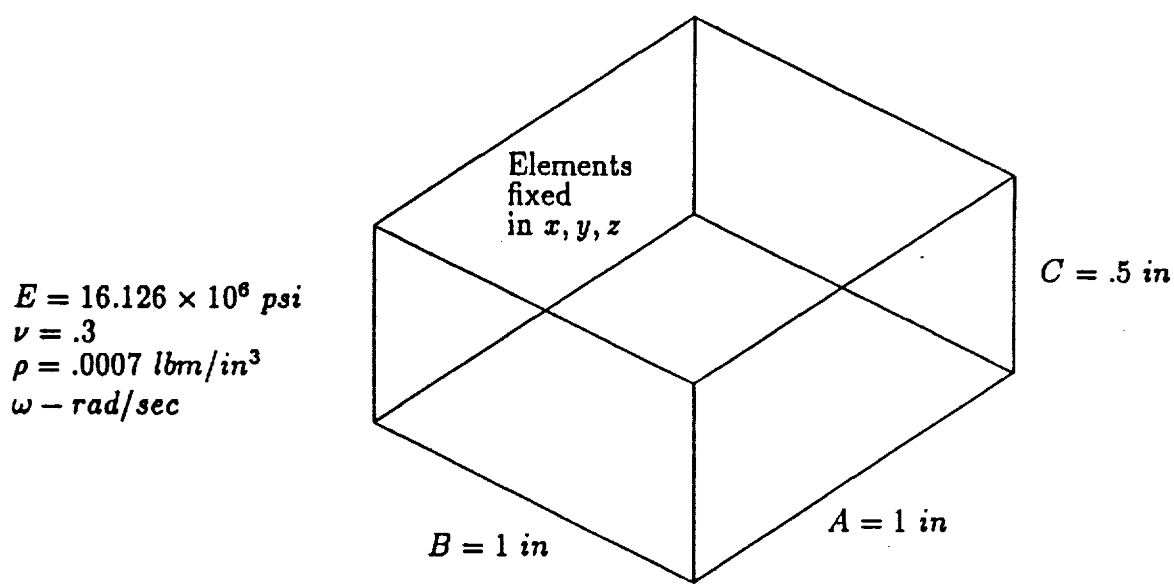


Fig. 7.2.1 Geometry for Calibration of **BEST3D** Natural Frequency Analysis

Model	Elements Along Side			Total Elements	Source Points
	A	B	C		
1	1	1	1	6	20
2	1	2	1	10	32
3	1	3	1	14	44
4	2	3	1	22	68

Table 7.2.1a **BEST3D** Modes for 1 X 1 X .2 Parallelepiped

Mode	Type	Ritz	Mesh 1	(%)	Mesh 2	(%)	Mesh 3	(%)	Mesh 4	(%)
1	EB	0.447	0.472	5.5	0.429	-4.0	0.435	-2.7	0.442	-1.2
2	SB	0.667	0.664	-0.4	0.666	-0.3	0.668	0.1	0.661	-0.8
3	T	0.788	0.887	12.5	0.829	5.1	0.820	4.0	0.788	0.0
4	L	1.596	1.625	1.8	1.620	1.5	1.618	1.4	1.602	0.4
5	EB	1.664	2.136	28.3	1.797	8.0	1.729	3.9	1.689	1.5
6	SB	1.774			1.836	3.5	1.789	0.9	1.775	0.1
7	T	2.220			2.552	14.9	2.448	10.3	2.285	2.9
8	EB	2.278					3.033	33.3	2.365	3.8
9	L	2.797							2.842	1.6
10	SB	3.068							3.249	5.9

Mode Type Identification — Frequency Parameter $= w\sqrt{\rho/E}$

EB - Easy Bending
SB - Stiff Bending
T - Torsion
L - Extension

Table 7.2.1b Convergence of **BEST3D** Natural Frequency Results

Calculation of Natural Frequencies for Twisted Plates [53]:

While the analyses discussed in the previous section demonstrated conclusively the ability of the boundary element method to calculate natural frequencies for three-dimensional bodies, the geometries involved were not of practical interest. In order to calibrate the method for more realistic geometries a number of twisted plate configurations were studied in considerable detail.

The geometries considered were used in the NASA sponsored Joint Research Effort on Vibrations of Twisted Plates. In this study four different combinations of plate length and thickness were studied, at each of five twist angles (equally spaced from 0^0 through 60^0). Specimens were machined and tested (at two laboratories) for each configuration. In addition, a large number of investigators made analytical predictions of the natural frequencies and mode shapes, mostly using finite element programs. The complete details of the experimental program are given in [57] and the various analytical results are collected in [58].

Because the three-dimensional boundary element method is directed primarily at the solution of truly three-dimensional problems, rather than the analysis of plates or shells, the most three-dimensional of the NASA plate geometries ($2in. \times 2in. \times .4in.$) was selected. A variety of **BEST3D** models was used, with that shown in Figure 7.2.2a finally chosen for detailed comparisons at all five twist angles.

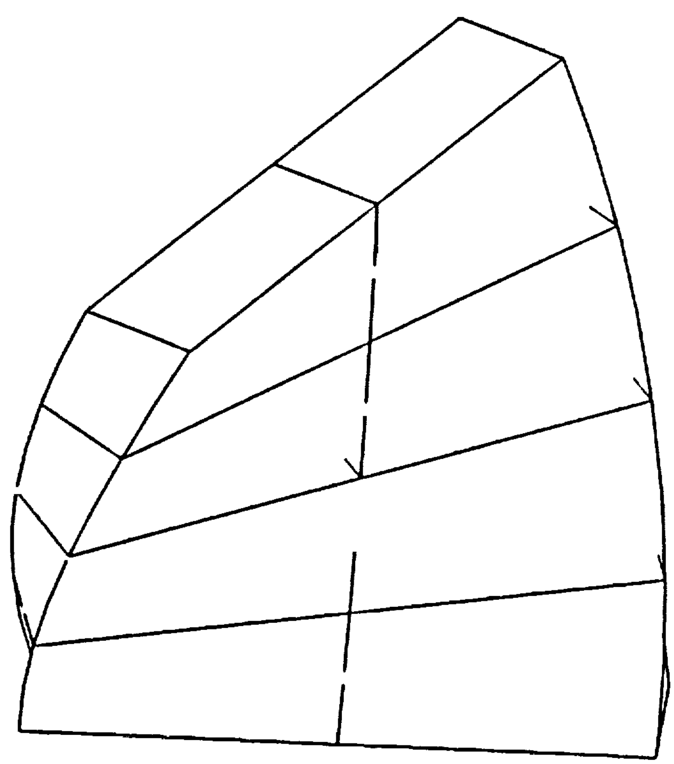


Fig. 7.2.2a Two Region **BEST3D** Model of 60° Twisted Plate

The twisted plate was modelled using two identical substructures. The calculated natural frequencies were compared with both experimental (when available) and finite element results. Four sets of finite element results were chosen for the comparison, since they gave the best results for this specimen configuration. The various analytical and experimental results considered and the designations given to them in the following figures are summarized in Table 7.2.2 In all of the figures the nondimensional frequency parameter, $\omega a^2\sqrt{\rho h/D}$, is plotted against twist angle, where ω is the natural frequency (rad/sec), a is the plate axial length, ρ is mass density, h is plate thickness and D is the plate flexural rigidity.

Designation	Type	Description	Degrees of Freedom
FEL	finite element	parabolic conoidal shell	252
FEM	finite element	quadrilateral thick plate	294
FEN	finite element	eight node solid isoparametric	660
FEO	finite element	sixteen node solid isoparametric	1872
BE	boundary element	BEST	288
EXA	experimental	NASA-LeRC	
EXB	experimental	Air Force Aero Propulsion Lab.	

Table 7.2.2 Analytical and Experimental Results used in Twisted Plate Comparison

As Figures 7.2.2b and 7.2.2c indicate, there is considerable scatter among the four sets of finite element results. The **BEST3D** results clearly fit within the range of the finite element results. **BEST3D** shows the closest agreement with the finite element results obtained using thick plate elements (method M) and sixteen node solid isoparametric elements (method O). The only analytical results available are the natural frequencies for the first bending and torsion modes for the untwisted plate. For these two modes **BEST3D** and the thick plate finite element method show closer and more consistent agreement than any of the other methods.

Experimental results are also shown in Figures 7.2.2b and 7.2.2c They are consistently lower than the analytical values for all modes. The lower experimental values are due to the compliance of the specimen base and fixturing. For the first two modes, both **BEST3D** and the thick plate finite element method correctly predict the variation of natural frequency with twist angle, although the finite element method overpredicts the variation for the first torsion mode.

It is clear that **BEST3D** predicts the twisted plated response at least as consistently as any of the finite element models. Final calibration of the method against experimental data required the construction of a **BEST3D** model containing at least a crude representation of the specimen base.

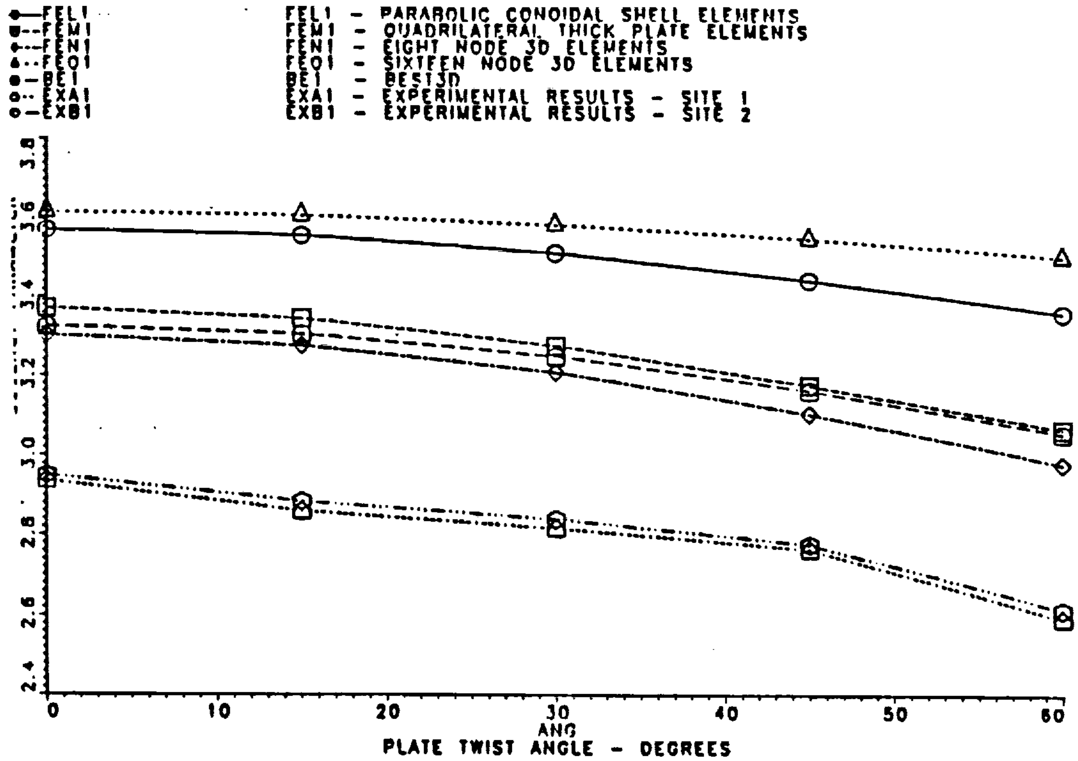


Fig. 7.2.2b First Bending Mode for $2 \times 2 \times .4$ Plate

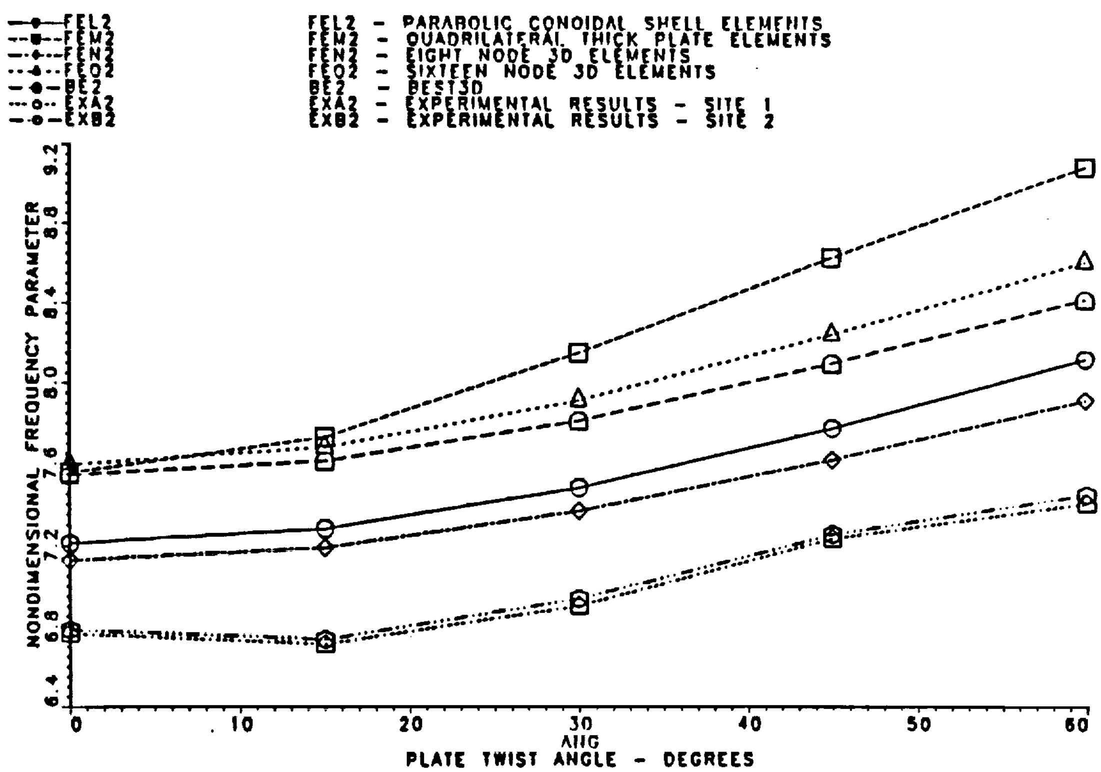


Fig. 7.2.2c First Torsion Mode for $2 \times 2 \times .4$ Plate

C5

The **BEST3D** model shown in Figure 7.2.2d was used to calculate the natural frequencies, at all five twist angles. The model consisted of three substructures, two for the plate and one for the specimen base. The lateral surface of the specimen base was completely fixed, to model the clamped condition intended in the experimental apparatus. As shown in Figures 7.2.2e and 7.2.2f, much better agreement with the experimental results was obtained, even with a relatively crude representation of the specimen base and clamping conditions.

Substructuring was found essential in order to obtain satisfactory results with surface meshes of reasonable density. This was true for the plate models, and was even more pronounced when the entire structure, including the base, was modelled. In part this simply reflects the well known advantages of substructuring in the static case, especially for the more complicated displacement and stress fields characteristic of bending. A further effect of substructuring in the natural frequency problem is the fact that it effectively introduces interior nodes into the problem. Since the $\rho\omega^2 u$ term cannot be exactly converted to boundary integrals, the introduction of interior sampling points (even indirectly, as boundary nodes on an interface) can be expected to produce significant improvement in the results.

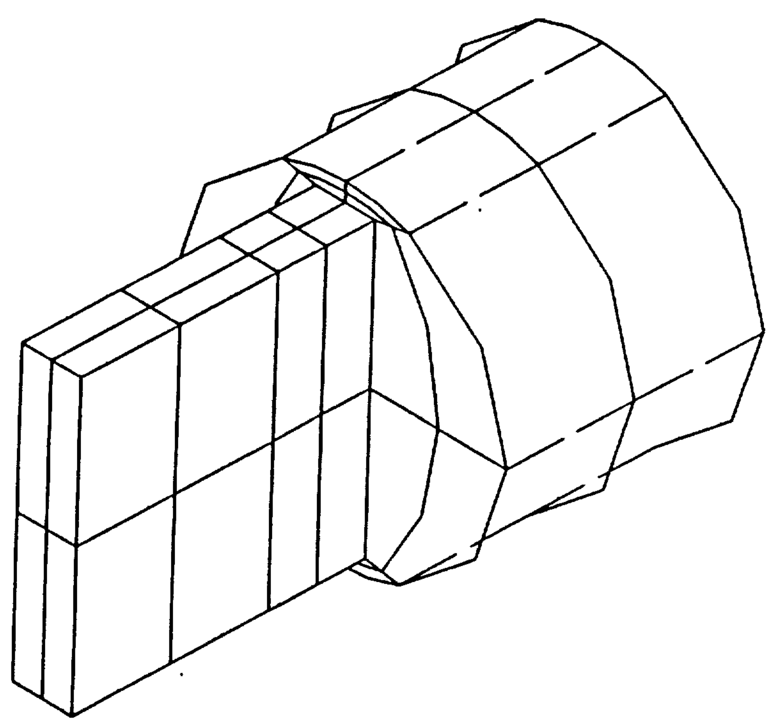


Fig. 7.2.2d **BEST3D** Model of $2 \times 2 \times .4$ Plate Including Base

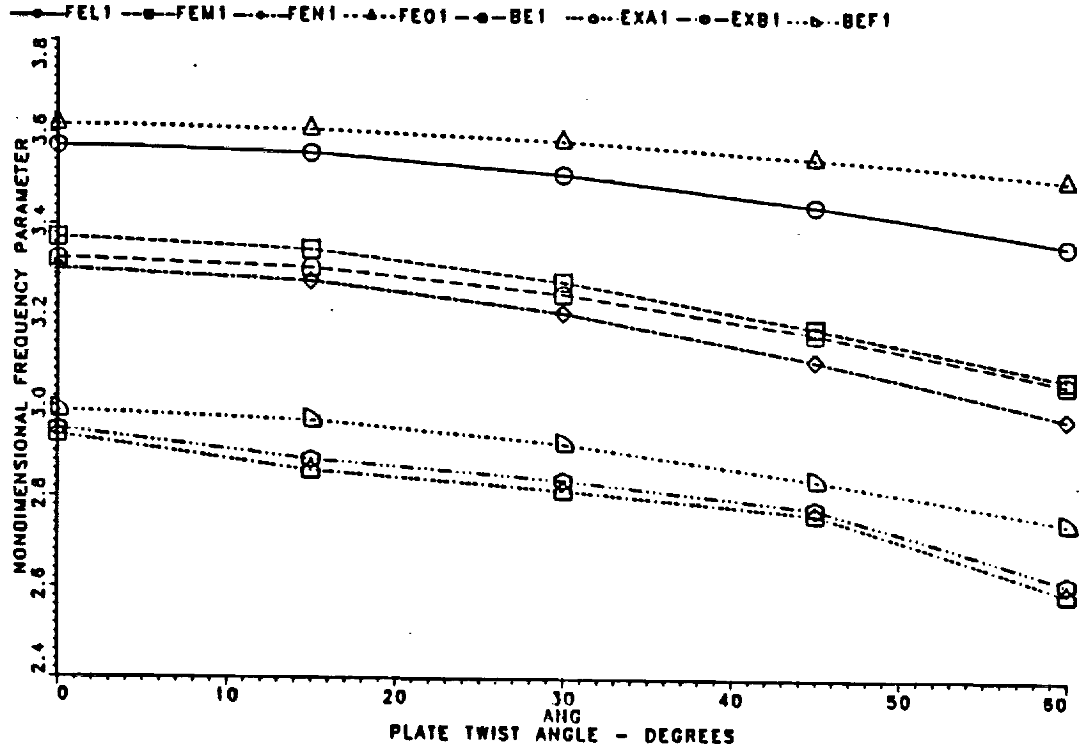


Fig. 7.2.2e Effect of Specimen Base on First Bending Frequency

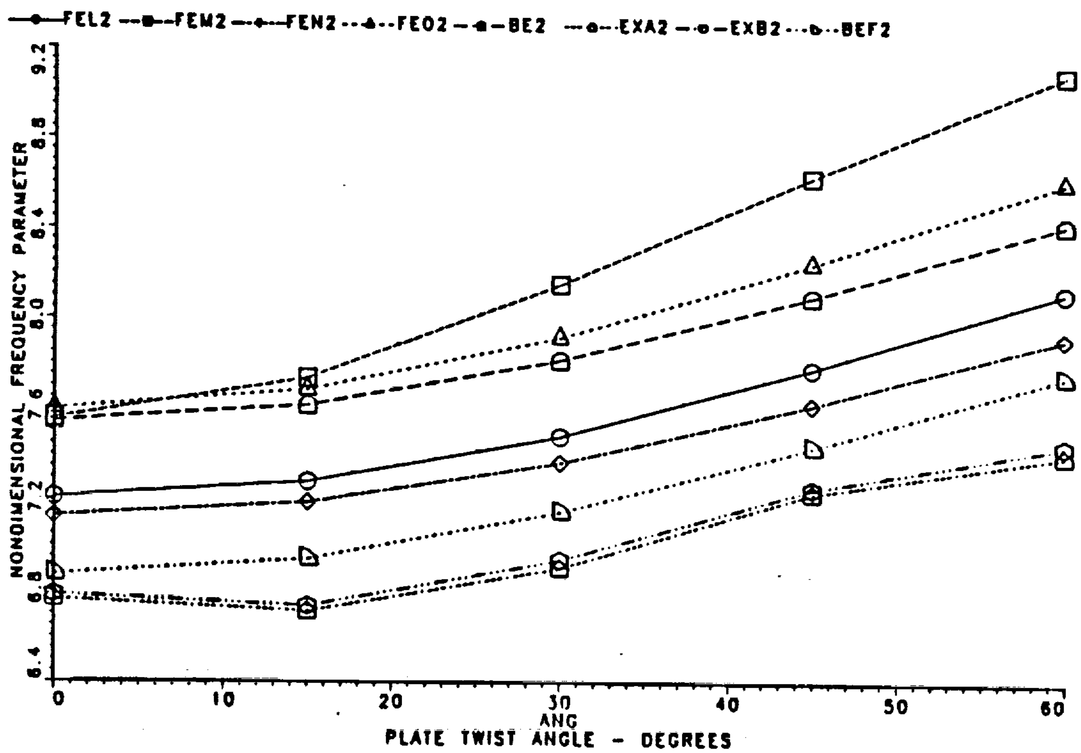


Fig. 7.2.2f Effect of Specimen Base on First Torsion Frequency

Comparison with Finite Element Results for an Automotive Component [53]:

The final example in this paper concerns the determination of the natural frequencies of an automobile crankshaft. A three-dimensional boundary element eigenvalue analysis was done using **BEST3D** . The boundary element mesh for this analysis is shown in Figure 7.2.3. A three-dimensional finite element analysis was also done, using a mesh in which the surface breakup was identical to the **BEST3D** model.

The results obtained from the two models are compared in Table 7.2.3a, where excellent agreement can be seen for the first, second and fourth modes (as determined by the **BEST3D** analysis). Computing times, on a CRAY XM-P, are summarized in Table 7.2.3b.

Mode	Finite Element	BEST
1	17400	18200
2	35800	34300
3	—	47500
4	66300	66300

Table 7.2.3a Natural Frequencies (Hz) for Automobile Crankshaft

	Finite Element	BEST
CPU (sec)	115	230
I/O (sec)	402	44

Table 7.2.3b Computing Times for Crankshaft Natural Frequency Analysis

Initial concern over a mode identified by **BEST3D** , but not by the finite element analysis, was resolved when a somewhat more refined finite element model gave a natural frequency at 47600 Hz, while leaving the first, second and fourth frequencies essentially unchanged. CPU time for the refined finite element analysis was very close to that for the **BEST3D** analysis.

In addition to providing further verification of the three-dimensional boundary element method free vibration capability, this example provides ample illustration of the desireability of having more than one analytical technique available for the solution of real engineering problems. In dealing with actual components the correct solution is not known *a priori*, expeimental results are often difficult to obtain and intuition may well fail. In such cases only the use of multiple analyses can give confidence in the calculated results.

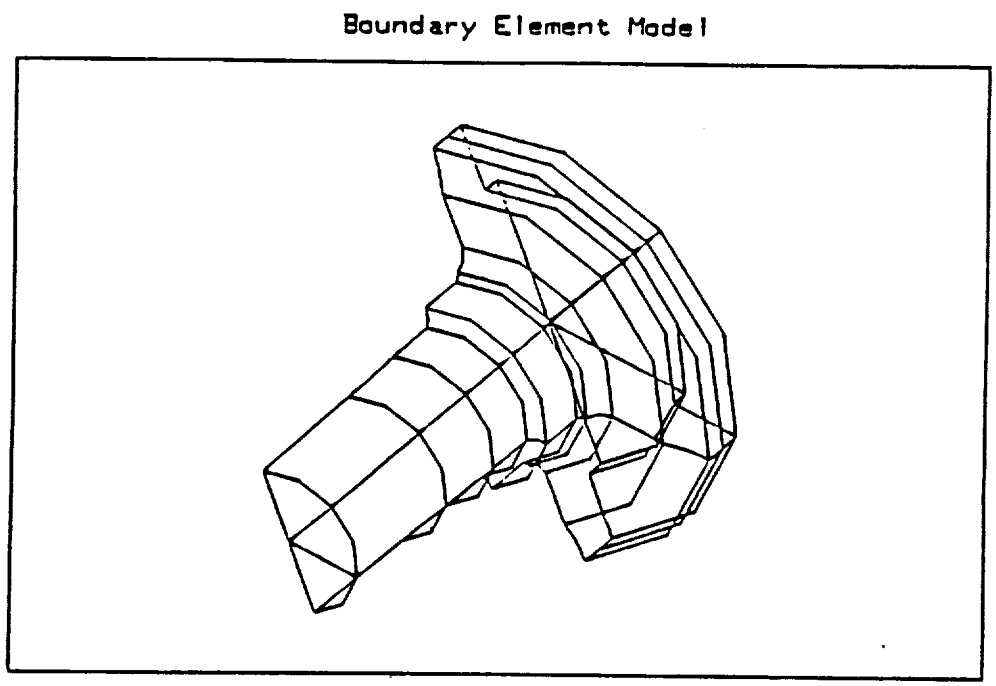


Fig. 7.2.3 **BEST3D** Model of Automobile Crankshaft

7.3 HEAT TRANSFER ANALYSIS

Tube and Disk Fin Heat Exchanger [42]:

This problem is the heat transfer analysis of a portion of a heat exchanger. Figure 7.3.1a shows a section of a thin fin attached to a thick tube carrying hot fluid. The tube (ID = 0.275 inch, OD = 0.375 inch) is modelled using 22 quadratic boundary elements. The thin (0.02 inch) fin is represented using 18 quadratic elements. Both regions have the same thermal conductivity of 25 in-lb/sec-in-°F. The tube carries a fluid with a temperature of 300°F and a convection coefficient of 60 lb/sec-in-°F. The fin is exposed to air at 100°F, with a convection coefficient of 15 lb/sec-in-°F. Since manufacturing processes cannot ensure perfect contact between the tube and the fin, various values of the thermal resistance at the disk-fin interface (0, 0.001, 0.005 and 0.02 in-sec °F/lb) were employed in a parametric study. Figure 7.3.1b shows the temperature distribution along a radial line through the tube and disk for the various values of thermal resistance. As expected, the temperature distribution becomes continous across the interface when the thermal resistance vanishes.

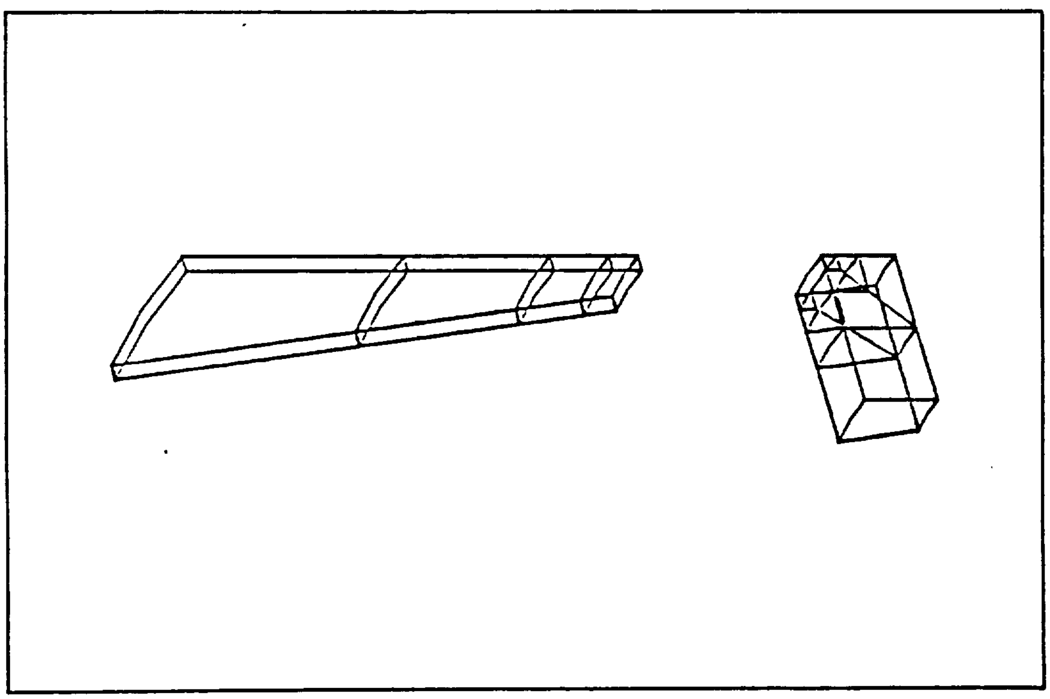


Fig. 7.3.1a Tube and disk fin heat exchanger - 2 GMR Boundary Element Model

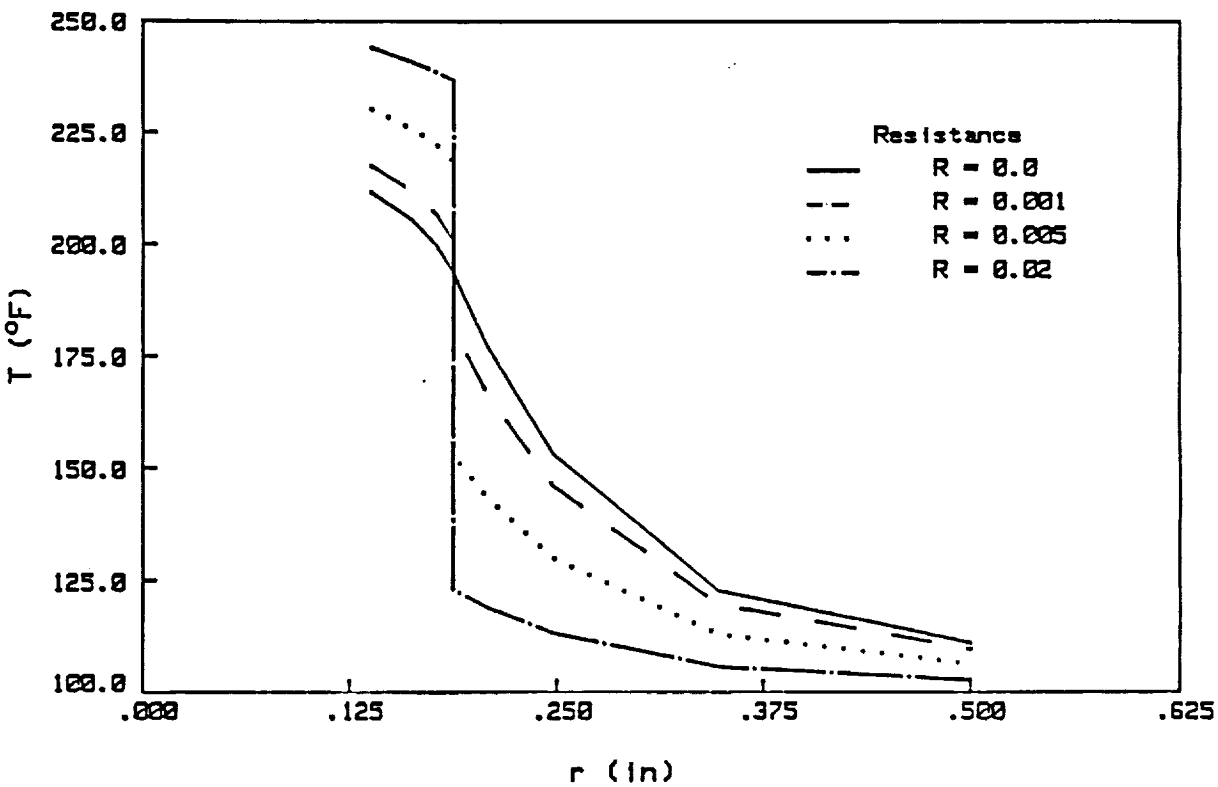


Fig. 7.3.1b Tube and disk fin heat exchanger
Results

Heat Transfer with Hole Elements [42]:

A new element class has been developed for the heat transfer version of **BEST3D** . These 'hole elements' are designed for the representation, without an explicit surface mesh, of cavities of cylindrical cross-section. The centerline of a single tube element is defined as a two- or three-noded isoparametric curve in three-space. Temperature and flux are allowed to vary along the centerline, but circumferential variation is ignored. Hole elements may be joined end-to-end, to improve definition of variation along the centerline or to allow representation of complex hole geometries.

In order to evaluate the accuracy and efficiency of the tube elements compared to explicit modelling, a model problem consisting of three cylindrical holes in a cube was solved. The tube length and radius are, x respectively, 80% and 1% of the cube dimension. The location of the cavities in the cube is shown in Figure 7.3.2a The front and back faces of the cube are maintained at 0 ° and 100°F. The cavity surface is subject to convection with a film coefficient of 5 lb/sec in °F and an ambient temperature of 200°F. The surface mesh for the cube consists of six quadratic boundary elements. In one analysis, each cavity was explicitly modeled using 48 linear boundary elements yielding a mesh with 150 elements, 422 geometric nodes and 152 source points. In the second analysis, each cavity was represented using 2 tube elements, giving a mesh of 6 surface elements, 6 tube elements, 35 geometric nodes and 35 source points. The analysis using tube elements required only 5% of the computer time needed for the explicit analysis.

The temperature variation along the tube for the two analyses is shown in Figure 7.3.2b. The tube element analysis almost exactly reproduces the much more expensive results of explicit modeling. Comparison of results on the surface of the cube shows agreement of unknowns (either temperature or flux) to better than 1%.

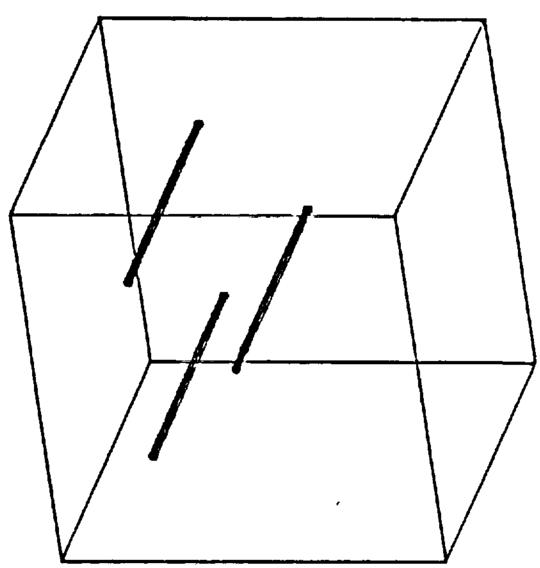


Fig. 7.3.2a Cube with embedded cylindrical holes
Boundary Element Model

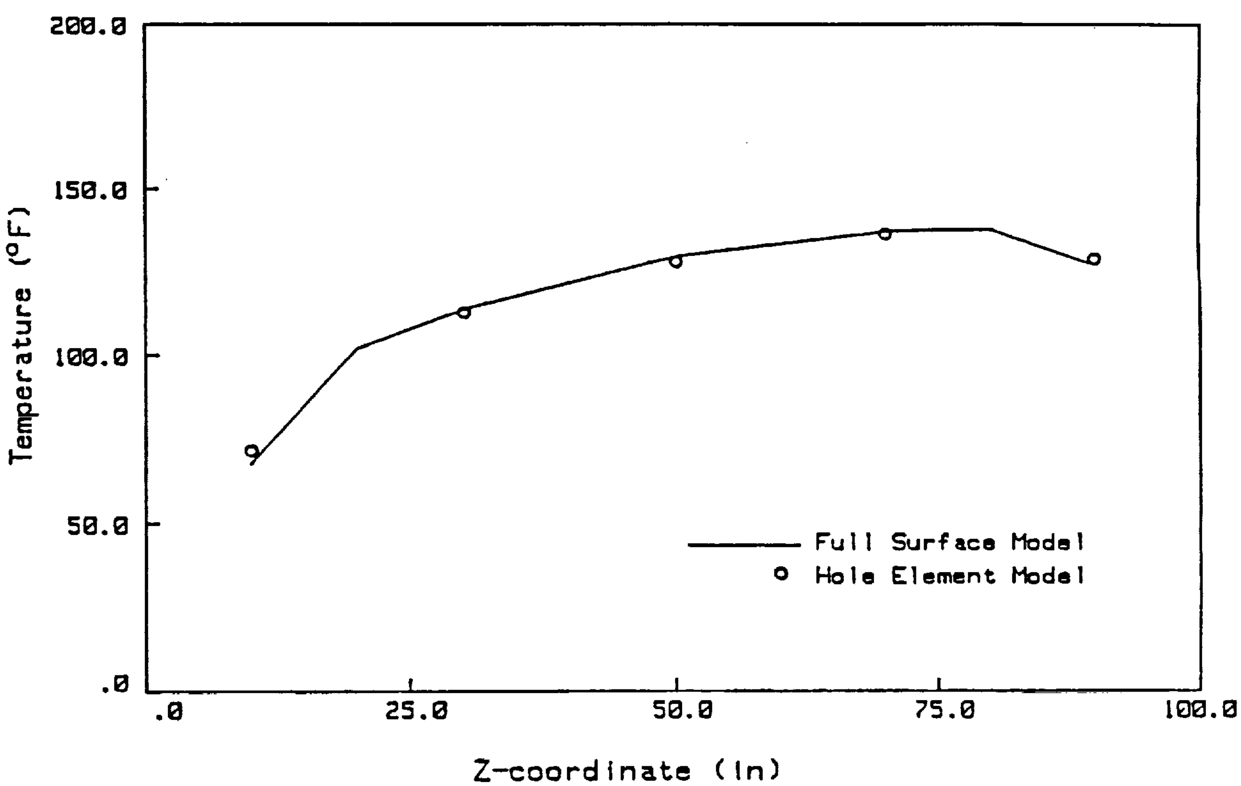


Fig. 7.3.2b Cube with embedded cylindrical holes
Temperature along central exclusion

Cooling of a Steel Sphere [59]:

A solid steel sphere of radius 1.5 in., initially at a uniform temperature of 100°F, is slowly cooled in 0° air by convection. The exact solution to this problem is given by Carslaw and Jaeger (1947) as

$$\theta(r,t) = \frac{2H\theta_o}{r}\sum_{n=1}^{\infty} e^{-c\alpha_n^2 t}\frac{R^2\alpha_n^2 + (RH-1)^2}{\alpha_n^2[r^2\alpha_n^2 + RH(RH-1)]} sinR\alpha_n sinr\alpha_n \tag{7.3.1}$$

where θ_o is the initial temperature, c is the diffusivity, R is the outer radius, and α_n are the roots of

$$R\alpha cot R\alpha + RH - 1 = 0. \tag{7.3.2}$$

Furthermore,

$$H = h/k \tag{7.3.3}$$

in which h is the convection film coefficient and k is the thermal conductivity. For positive H, all the roots of (7.3.2) are real. To complete the problem specification, let

$c = 0.0205 in.^2/sec.,$

$k = 25 in. - lb/sec.in.^{o}F,$

$h = 1.25 in. - lb/sec.in.^{o2}F.$

Three levels of mesh refinement are considered to study convergence of the boundary element solution. In each case, only the positive octant of the sphere is modeled, while symmetry constraints are imposed. Figures 7.3.3a - 7.3.3c illustrate the boundary element idealizations employed in this analysis. Note that the plotting package, used to produce these plots, connects all nodes by straight lines, however, within **BEST3D** , a quadratic variation of geometry is utilized for each surface element. The results from **BEST3D** , for a time step of 1.25 sec., are compared with the exact solution in Figure 7.3.3d for points on the surface and at the center of the sphere. Correlation is good, even for the three element model which never deviates from the exact solution by more than 2°F. Dramatic improvement occurs for the four element mesh, but then diminishing returns are evident upon further refinement. However, the trend toward convergence is apparent from the graph.

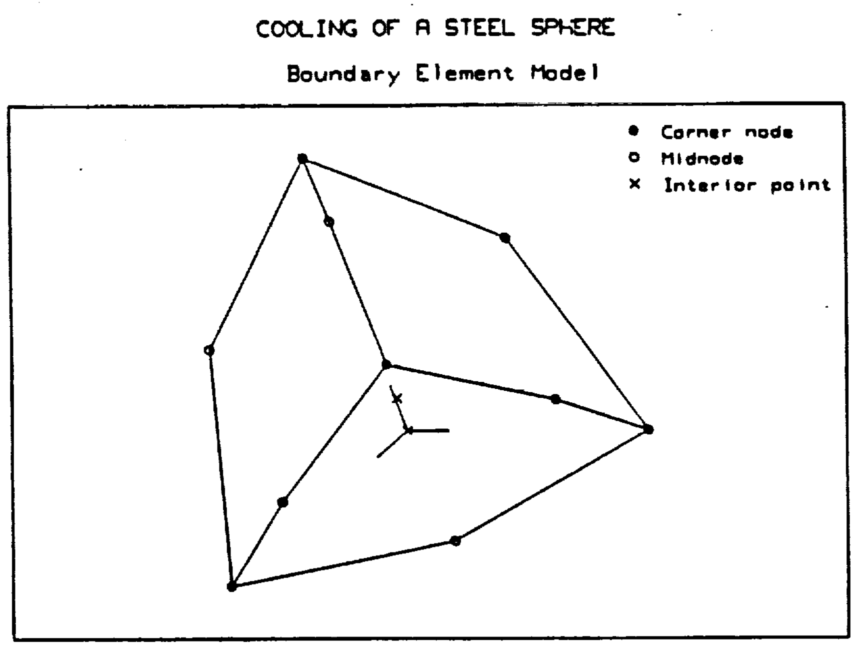


Figure 7.3.3a

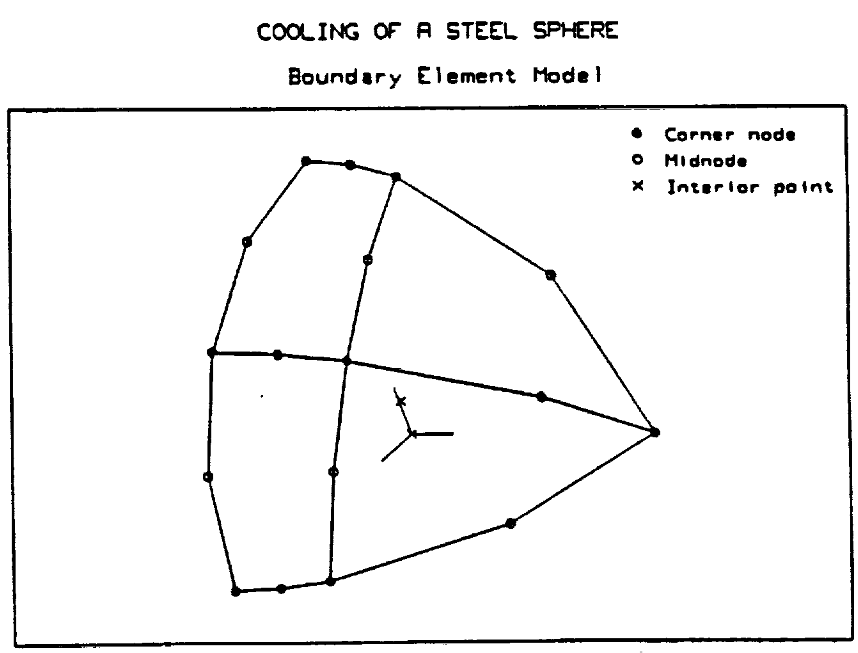


Figure 7.3.3b

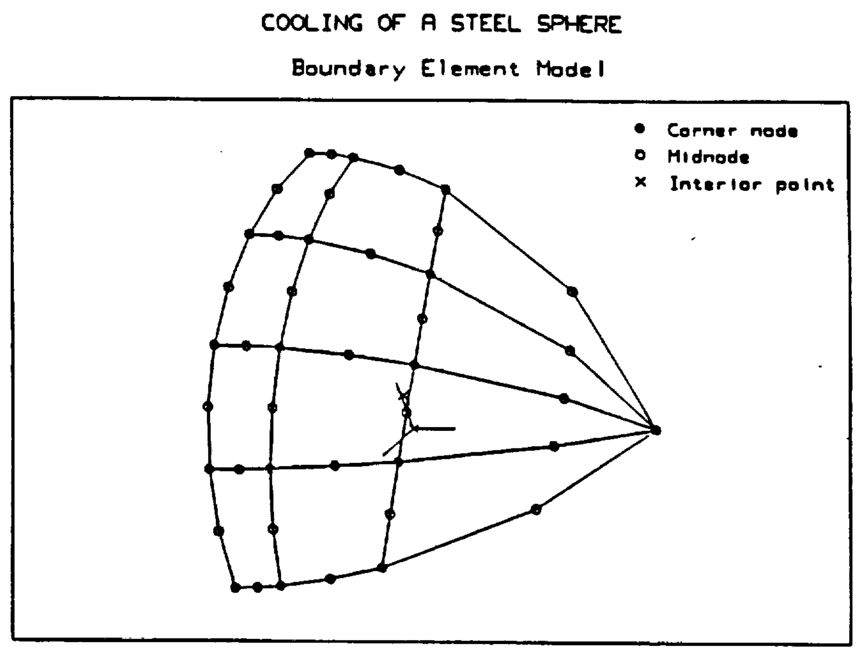


Figure 7.3.3c

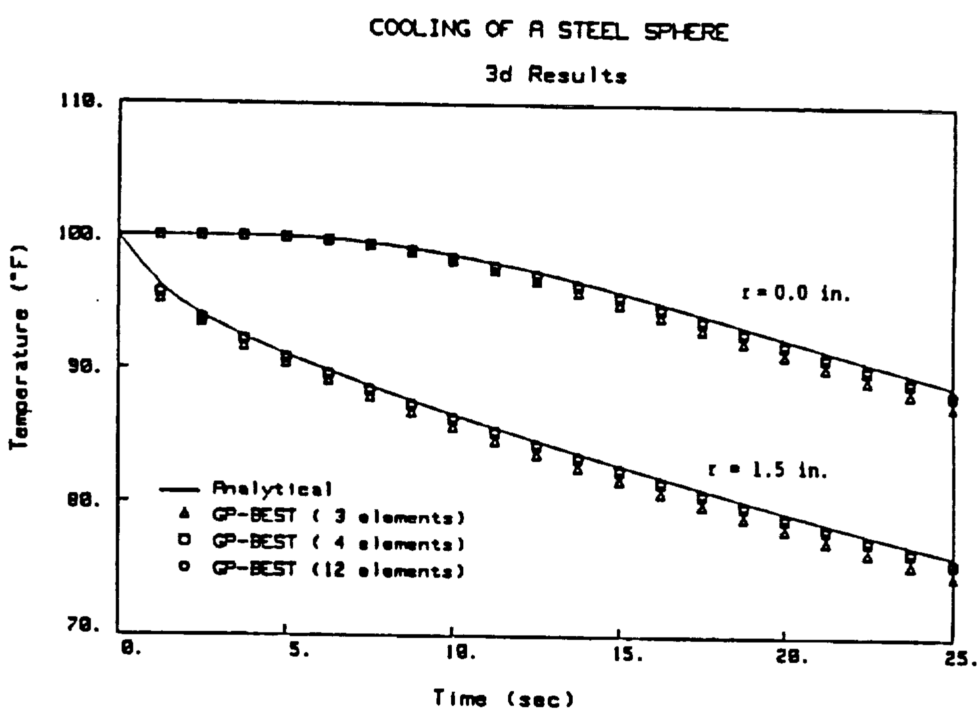


Figure 7.3.3d

Injection Mold [42]:

In this problem, the steady-state temperature distribution in an injection mold is investigated. The specific mold under consideration is used to manufacture semicircular thermoplastic tanks. The tanks have an inner radius of 1.0in., a thickness of 0.25in., and an overall lenght of 5.5in. Meanwhile, the mold itself is quite simple. It consists of two steel components, a core and a cavity, with combined outer dimensions of 15in. × 20in. × 20in. Several 0.25in. dimeter cooling lines, running parallel to the lenght of the tank, are provided in both components.

The boundary element model of the positive quadrant of the mold is depicted in Figure 7.3.4a. In that diagram, the core and cavity are shown individually as separate GMR's, which are viewed from different vantage points. Thirty-two quadratic surface elements model the core, while the cavity is discretized into twenty-nine elements. Additionally, four quadratic hole elements are employed to represent each cooling line. The location of these cooling lines are defined in the cross-sectional slice displayed in Figure 7.3.4b.

Convective heat transfer is assumed on all exposed surfaces. Specifically, on the surface of the tank h = 25in.-lb./in.^{2}sec$^\circ$ F and $\theta_{amb} = 450^\circ$, while for the cooling lines h = 50in.-lb./in.^{2}sec$^\circ$ F and $\theta_{amb} = 120^\circ$. A much lower convection coefficient (h = 0.1in.-lb./in.^{2}sec$^\circ$ F) is assumed on the outer mold surfaces which are in contact with 100°F ambient air. Also, the thermal resistance of 0.01in.^{2}sec.°F/in.-lb. is imposed across the imperfect core to cavity interface. The conductivity of steel is taken as 5.8in.-lb./sec.in.°F.

The resulting steady-state temperature contours are displayed in Figure 7.3.4c. Notice that the highest temperatures occur near the crown of the tank, while the largest thermal gradients appear near the foot. Also, as is evident from Figure 7.3.4d, there are significant temperature differences between the inner (core side) and outer (cavity side) surfaces, which may lead to the development of residual stresses, and subsequently, warpage in the molded tank.

In order to provide a more uniform surface temperature profile, typically, the cooling lines must be repositioned and/or resized. Each design iteration can be examined quite efficiently with the present boundary element approach. In fact, once the outer surface of the mold is defined, at each iteration, the mold designer is only required to position the cooling hole centerline nodes, and provide the appropriate boundary conditions. On the other hand, it is difficult to visualize the utility of a finite element or finite element difference approach, which would require the generation of an extremely complicated mesh for a three-dimensional solid with an irregular outer boundary and a number of embedded holes.

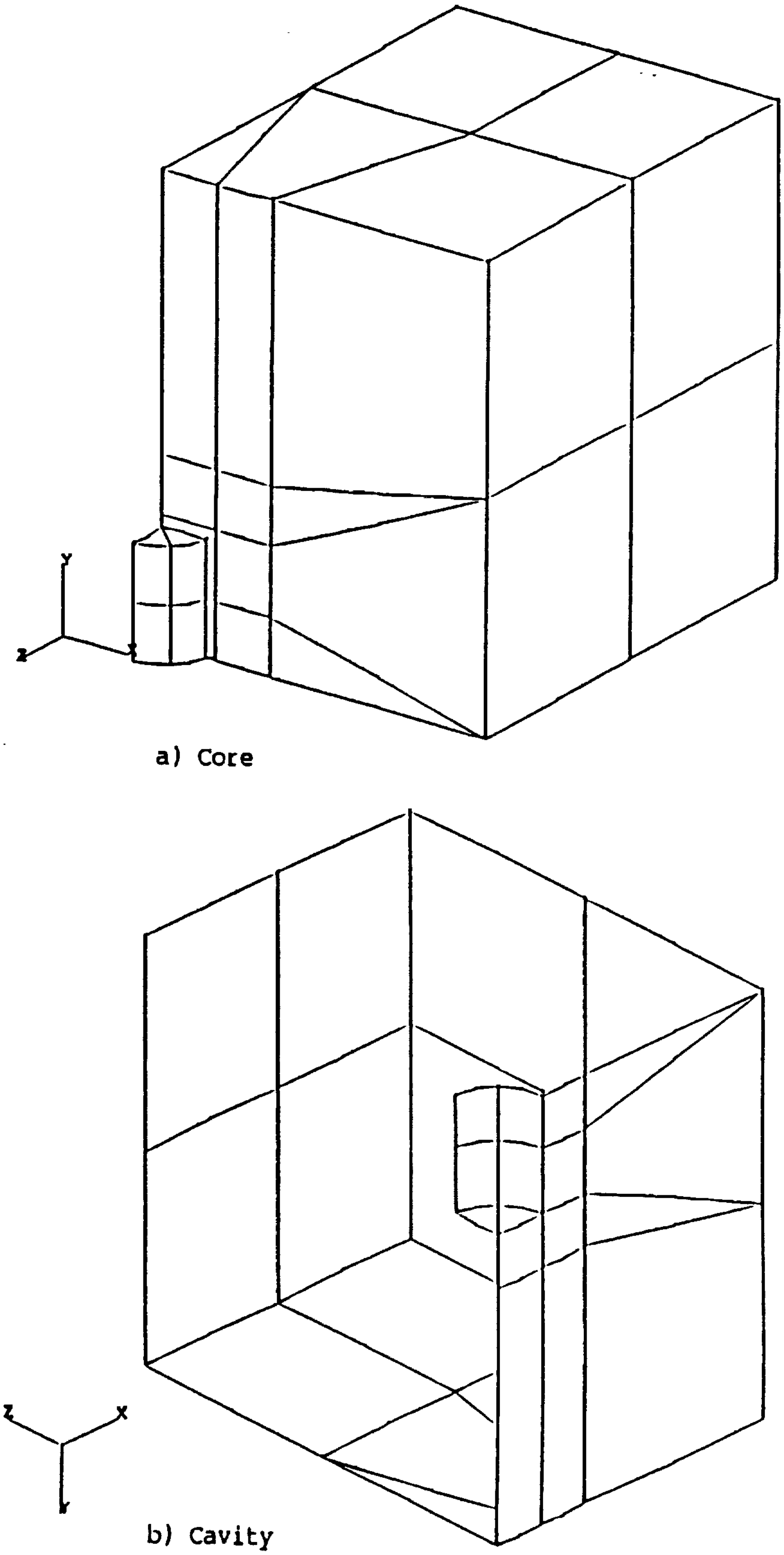


Figure 7.3.4a Injection Mold - Boundary Element Model

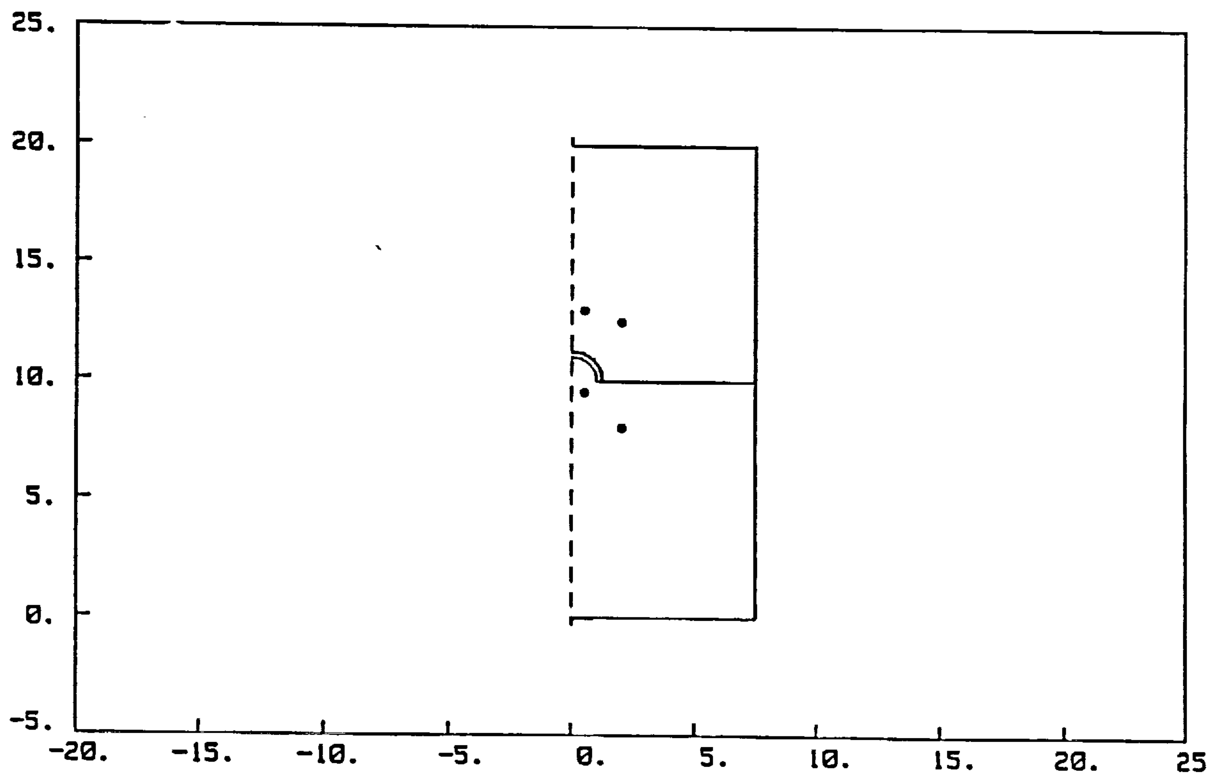


Figure 7.3.4b Cross-section - Injection Mold

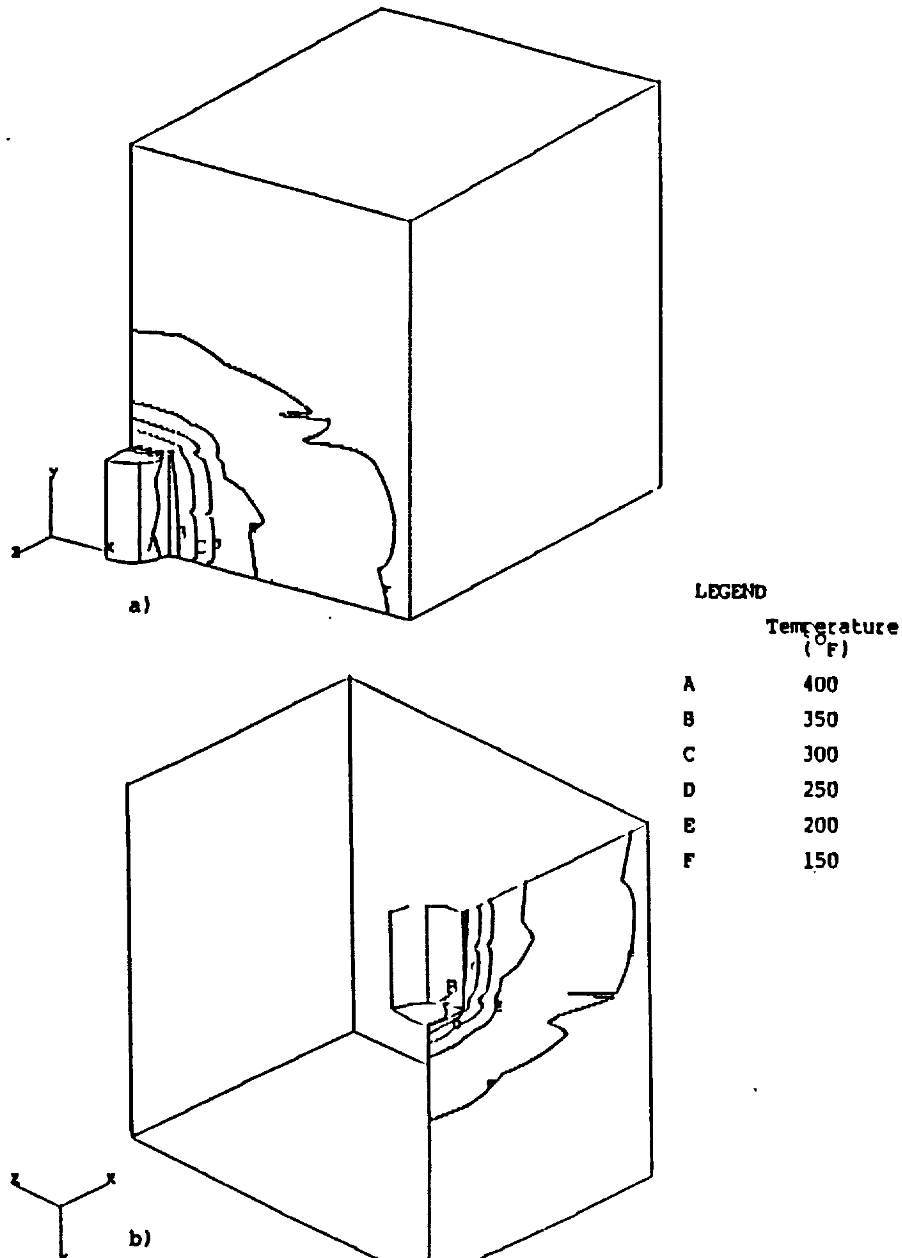


Figure 7.3.4c Temperature Contours - Injection Mold

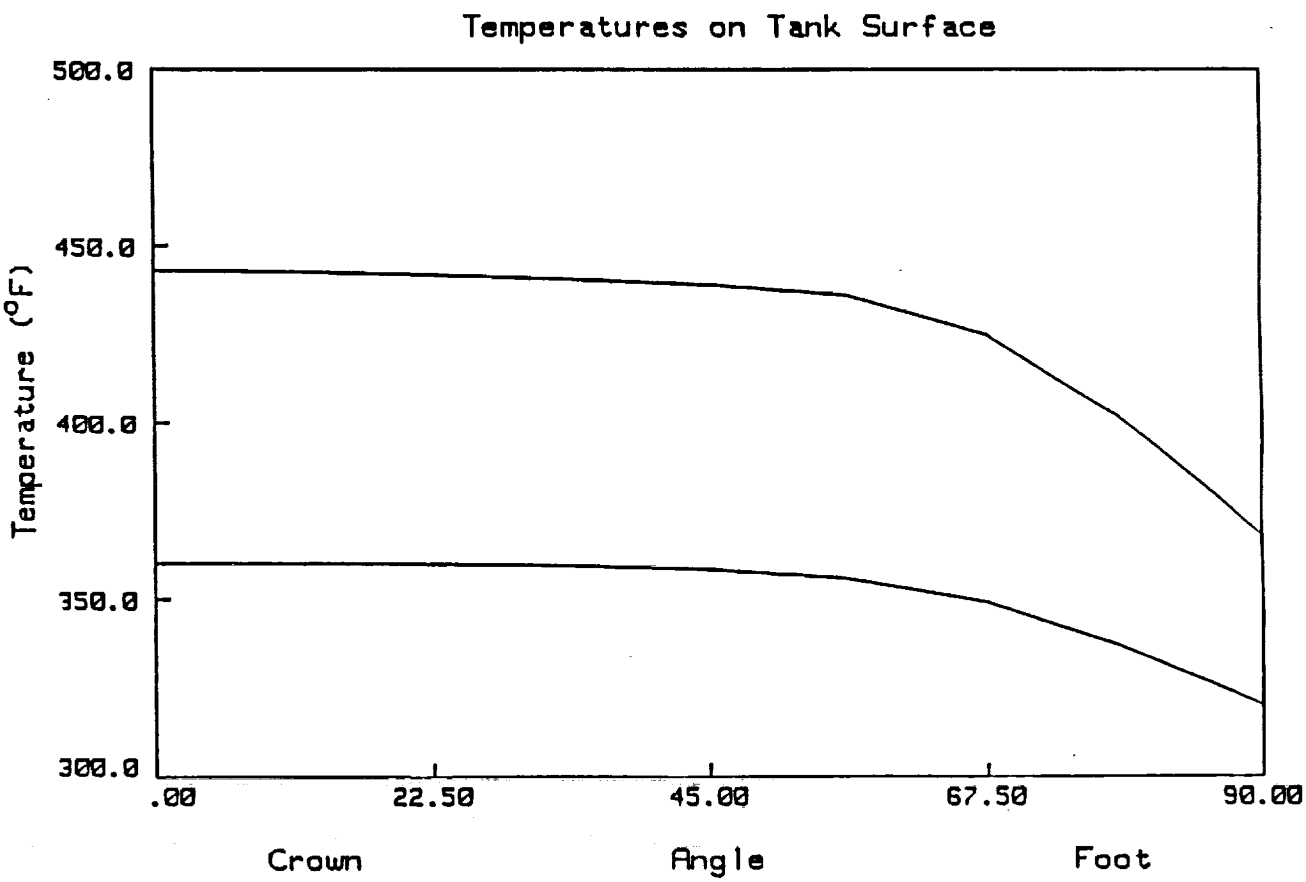


Figure 7.3.4d Temperatures on Tank Surface - Injection Mold

Thermal Response of a Turbine Blade [33]:

In this example, the transient thermal response of a turbine blade is examined during a warm startup. The boundary element surface model of the component is shown in Figure 7.3.5a. The model is composed of three GMR's with a total of 128 elements and utilizes symmetry about the x=0 plane.

The initial state is first determined with a steady-state thermal analysis, conducted under the following conditions. The exposed portion of the blade, as indicated in Figure 7.3.5a, is heated via convection by a gas at 1170°C. Convection coefficients are a function of position along the surface of the blade and are plotted versus the chord lenght in Figure 7.3.5b. Meanwhile, the lower blade and cylinder edges are heated by a 500°C gas and the cylinder base is exposed to a 300°C fluid. In both of these latter instances, a relatively low convection coefficient of 0.000050 W/mm^{2}C is specified. The tip of the blade is completely insulated. Additionally, the thermal material properties are specified as follows :

$K = 0.0216$ W/mmC
$C = 6.14$ mm^2/sec.

The resulting steady-state temperature distribution is presented in Figure 7.3.5c. Peak temperatures occur near the tip, however the highest thermal gradients appear near the blade-to-cylinder junction. Next, the turbine blade is analyzed under transient conditions. For this case, the gas temperatures vary according to the profiles depicted in Figure 7.3.5d. Convection coefficients remain unchanged from the steady-state values detailed previously. The thermal response of the component, as determined with a time step of 0.25 sec., is provided at various times in Figures 7.3.5e-h. Notice that the blade cools dramatically during the initial two seconds, and produces relatively sharp thermal gradients, particularly near the leading edge. Then, as the gas temperature surrounding the blade continues to rise, the peak gradients are again shifted toward the blade-to-cylinder junction. Of course, all of these results could next be used as input for a thermal stress analysis.

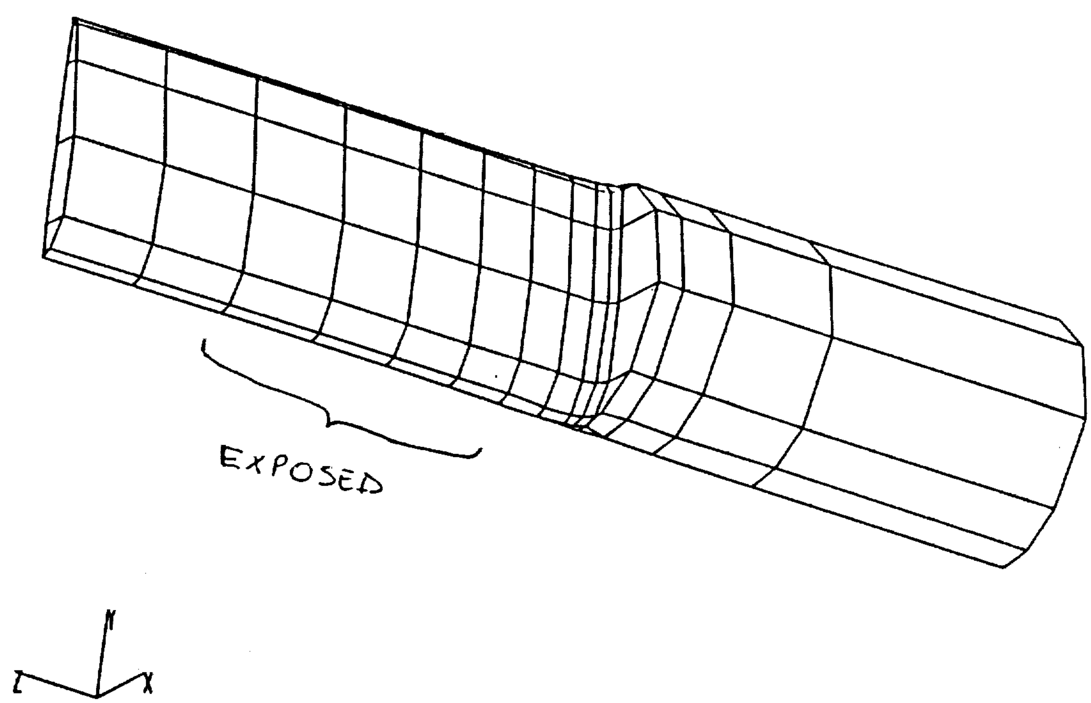


Fig. 7.3.5a Thermal Response of a Turbine Blade
Boundary Element Model

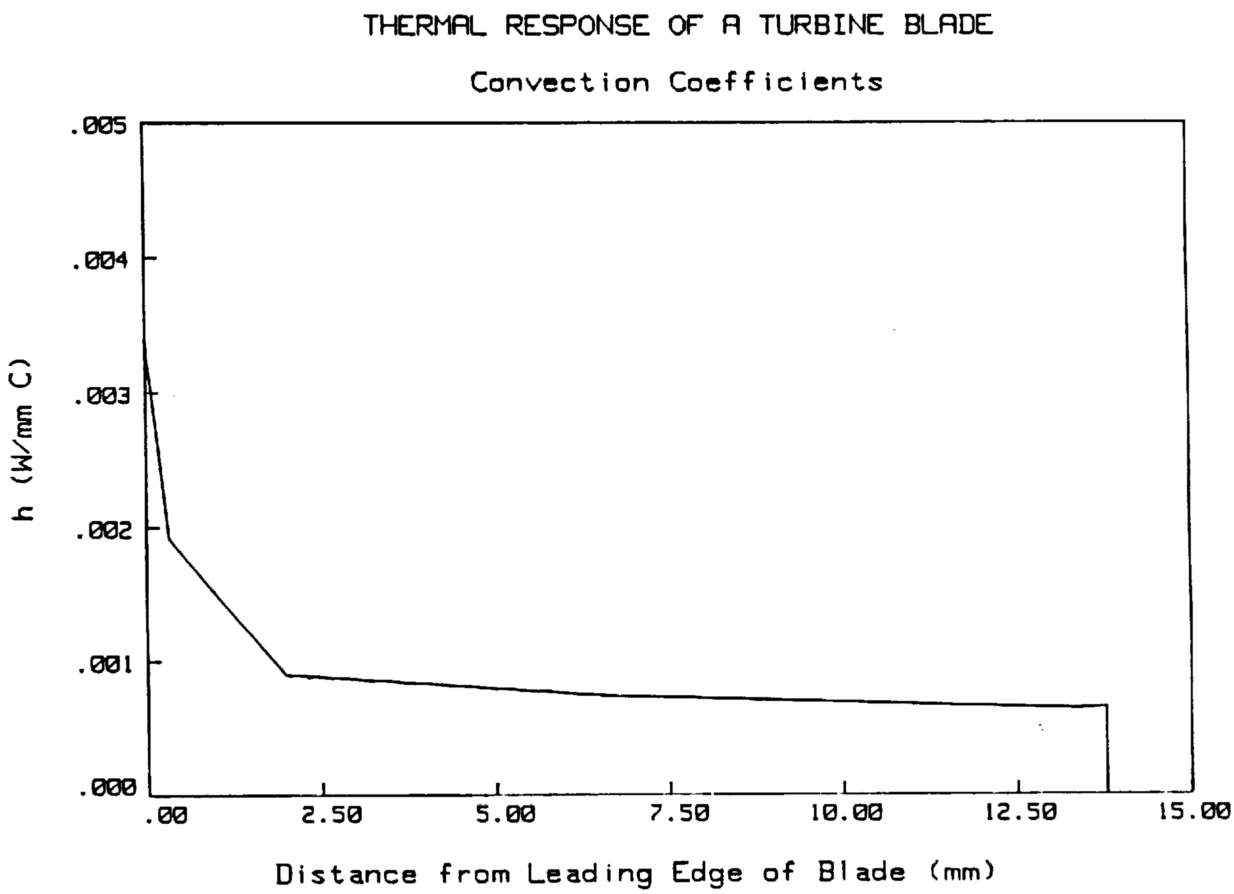


Fig. 7.3.5b Thermal Response of a Turbine Blade
Convective Coefficients

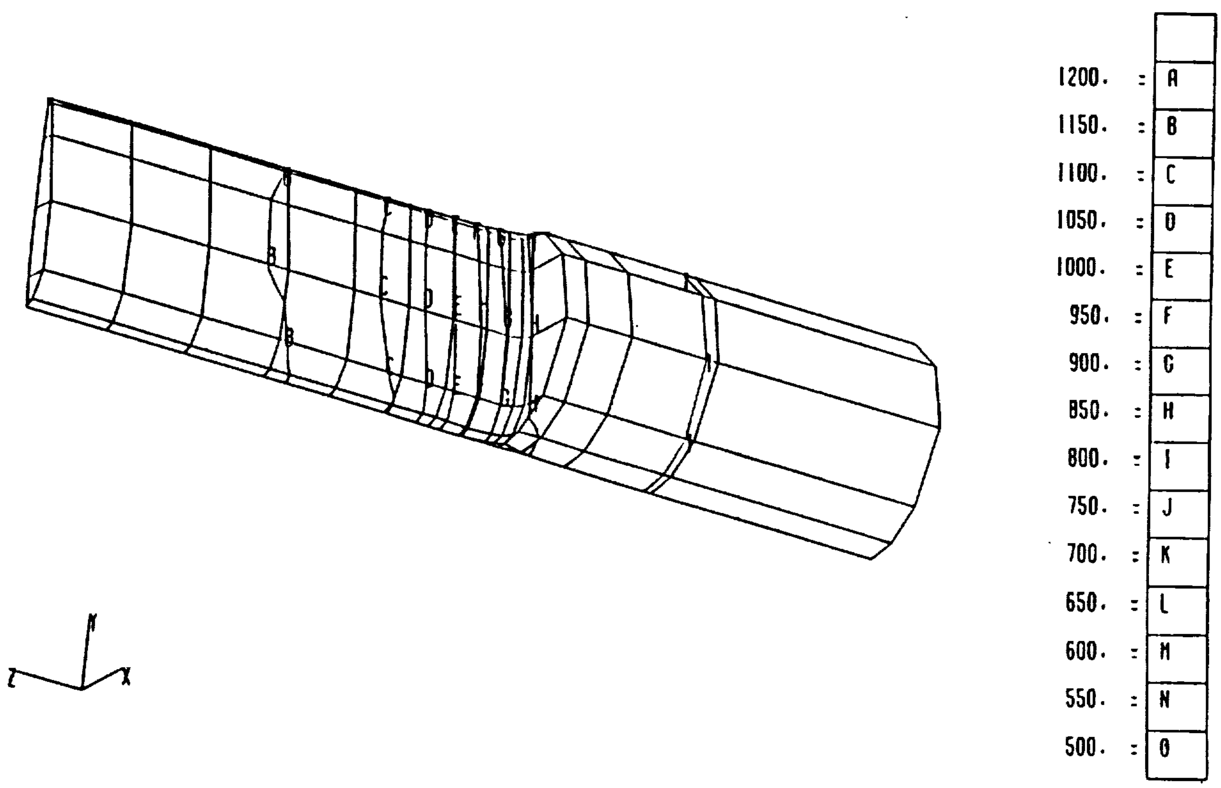


Fig. 7.3.5c Thermal Response of a Turbine Blade
Steady-state Temperatures

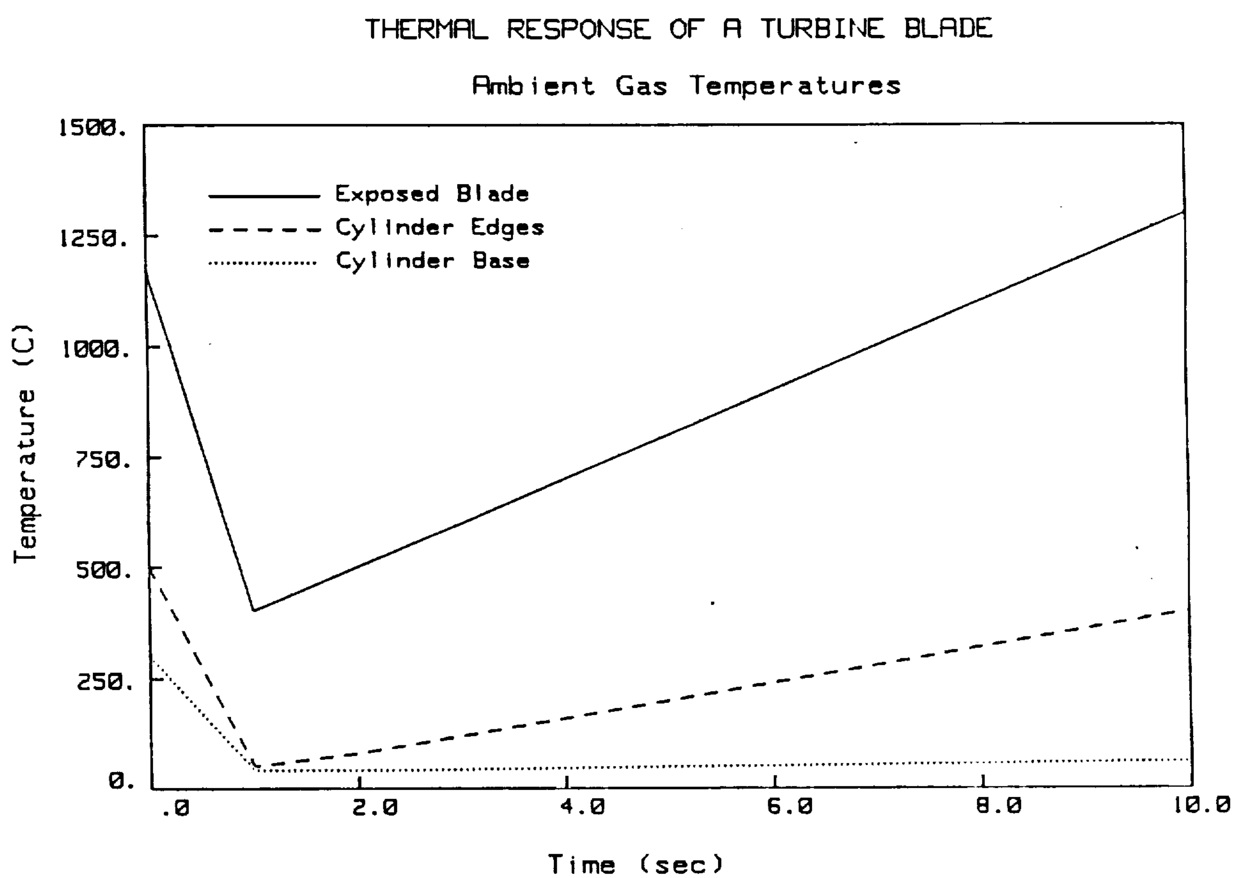


Fig. 7.3.5d Thermal Response of a Turbine Blade
Ambient Gas Temperatures

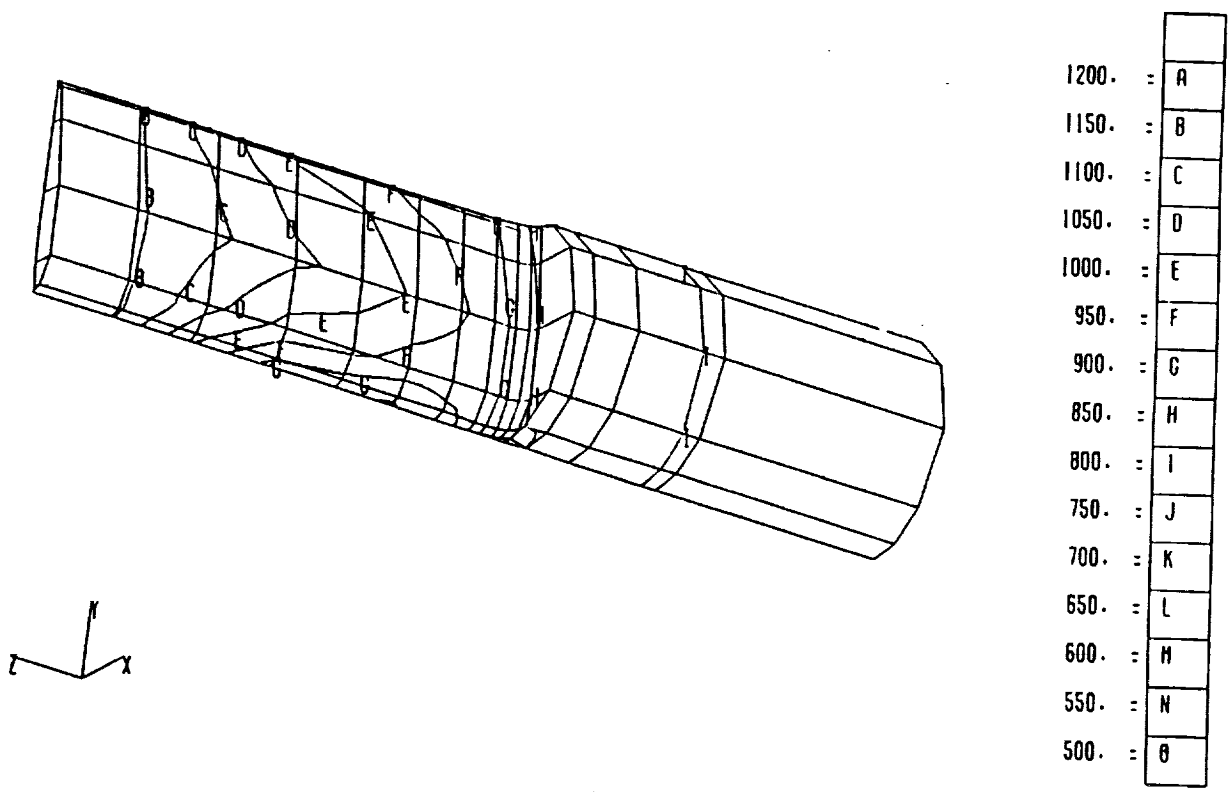


Fig. 7.3.5e Thermal Response of a Turbine Blade
Transient Temperatures at t = 1 sec.

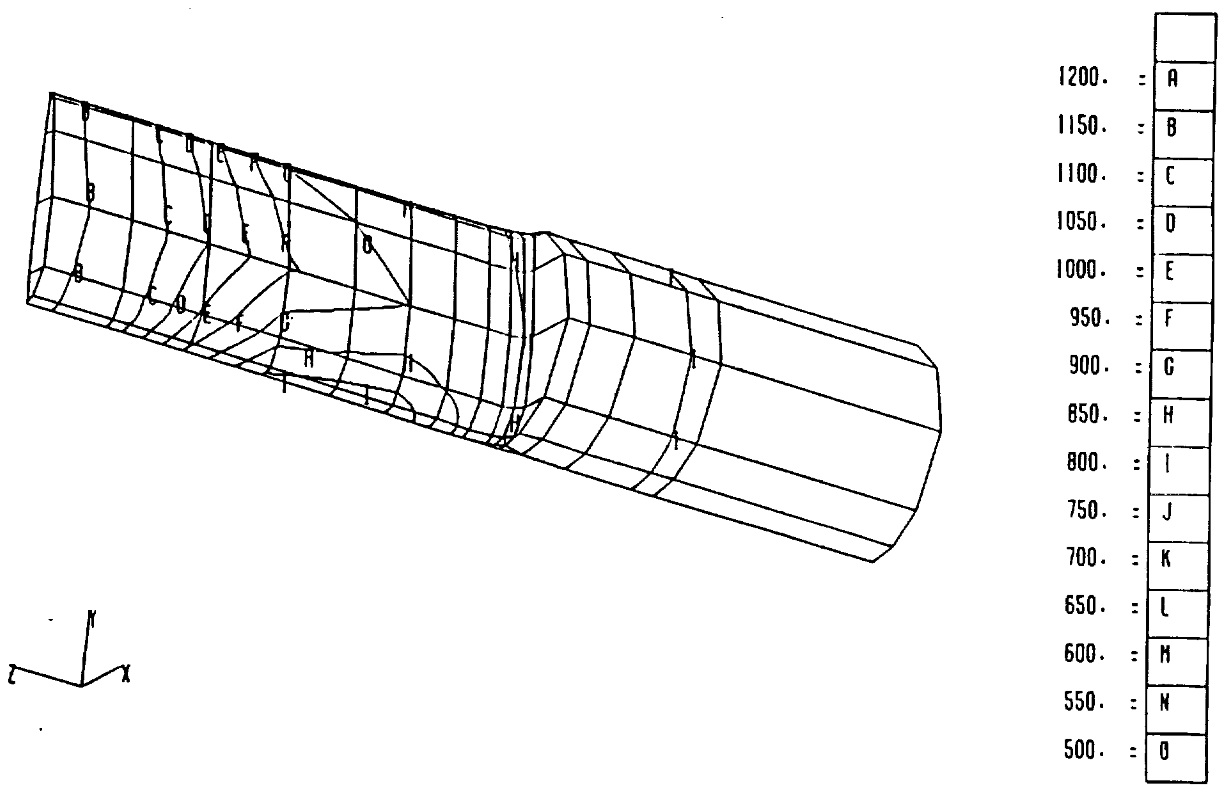


Fig. 7.3.5f Thermal Response of a Turbine Blade
Transient Temperatures at t = 2 sec.

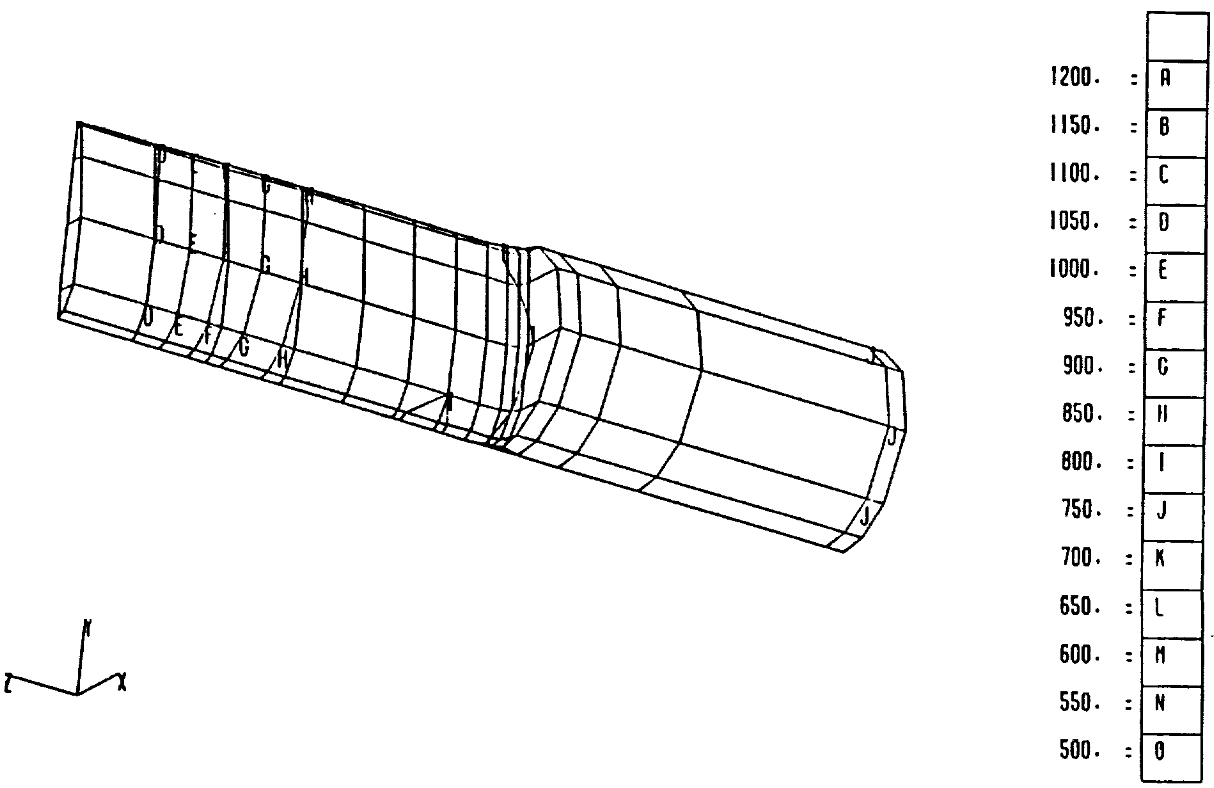


Fig. 7.3.5g Thermal Response of a Turbine Blade
Transient Temperatures at t = 5 sec.

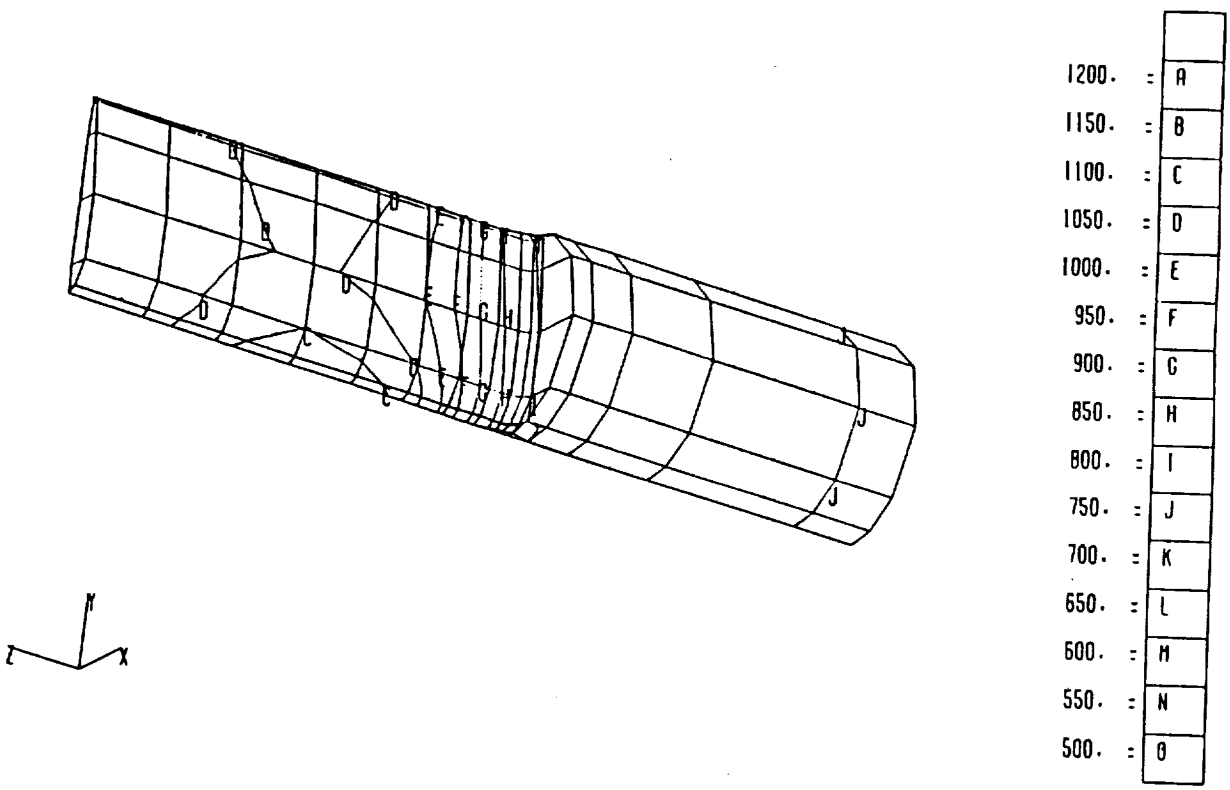


Fig. 7.3.5h Thermal Response of a Turbine Blade
Transient Temperatures at t = 10 sec.

Dynamic Expansion of a Spherical Cavity [26,28]:

The problem of the dynamic radial expansion of a spherical cavity within an infinite solid was used as a test problem to validate both the Laplace transform transient dynamic analysis and the time domain transient analysis. The radius of the cavity was assumed to 212 inches, and the material parameters were Young's modulus = 8.993 x 106, = 0.25, mass density = 0.00025. A radial pressure of 1000 psi was suddenly applied and maintained.

Three different discretizations used to analyze this problem are shown in Figure 7.4.1a. Figure 7.4.1b shows the Laplace transform and the transient time domain results for mesh 2 compared with the exact analytical solution. In general, the numerical results are in good agreement with the analytical solution. These is some oscillations in the results of the Laplace transform solution which becomes progressively worse beyond t = 0.0056 secs. The transient time domain solution, on the other hand, remains stable and reaches the expected static solution at larger times. The numerical errors in the displacement-time response characteristics for the meshes 1, 2 and 3 were about 15%, 9% and 6%, respectively.

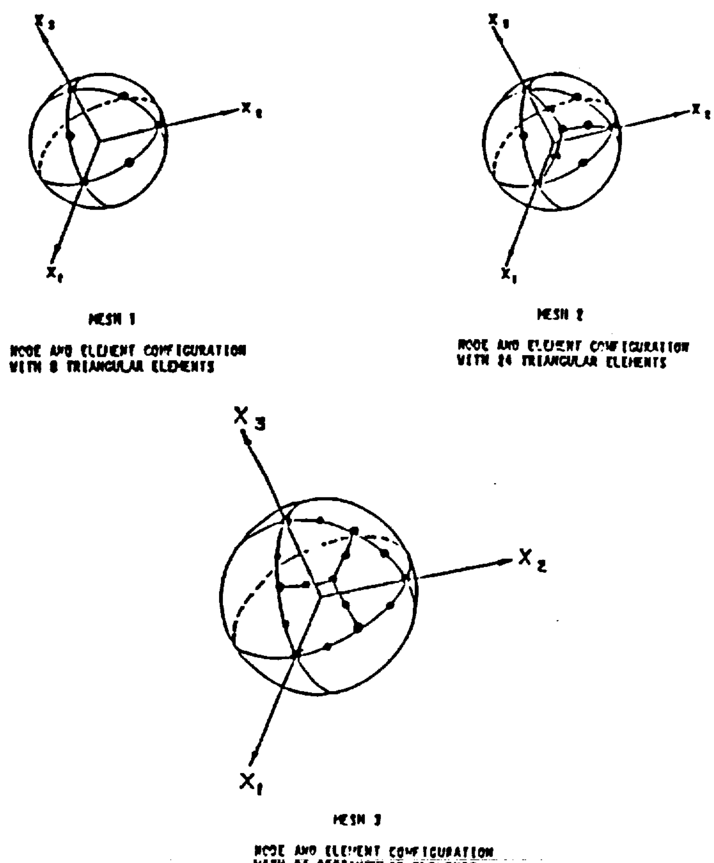


Fig. 7.4.1a Discretization of the surface of a spherical cavity

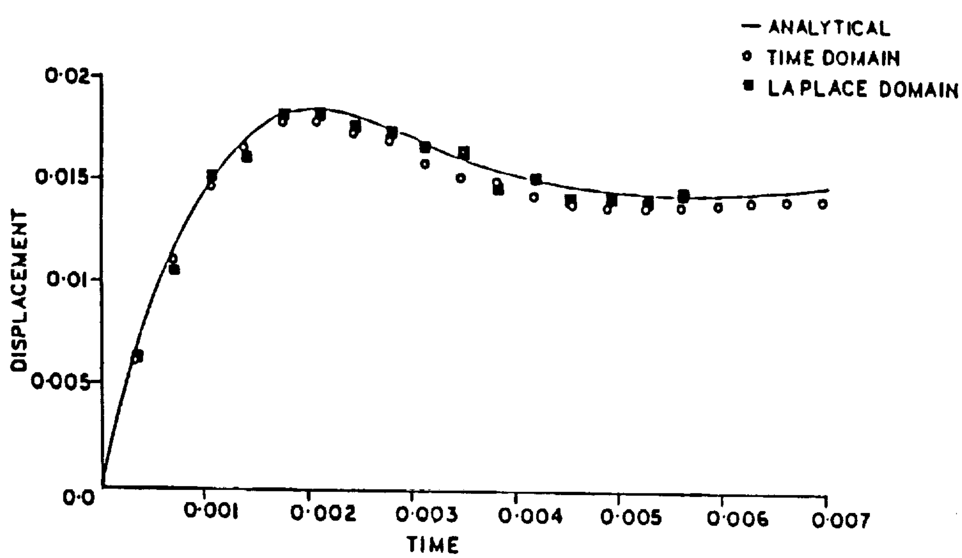


Fig. 7.4.1b Comparison of the Laplace transform and the time domain transient results with the analytical solution.

A Column Subjected to a Transient End Load [26,27]:

A column (2 x 2 x 8) held along its sides by lubricated rollers and supported on a frictionless base is subjected to a suddenly applied pressure of 1000 at the free end. The material properties are Young's modulus = $1\text{x}10^5$ and Poisson's ratio = 0. Figure 7.4.2 shows the discretization and numerical results for two time step sizes compared with the exact analytical solution for one-dimensional stress wave propagation. Although the numerical results are in good agreement with the analytical solution it is clearly very difficult to reproduce the sharp jump in the stress as the disturbance reaches the point initially and when the reflected stress wave returns to the same location.

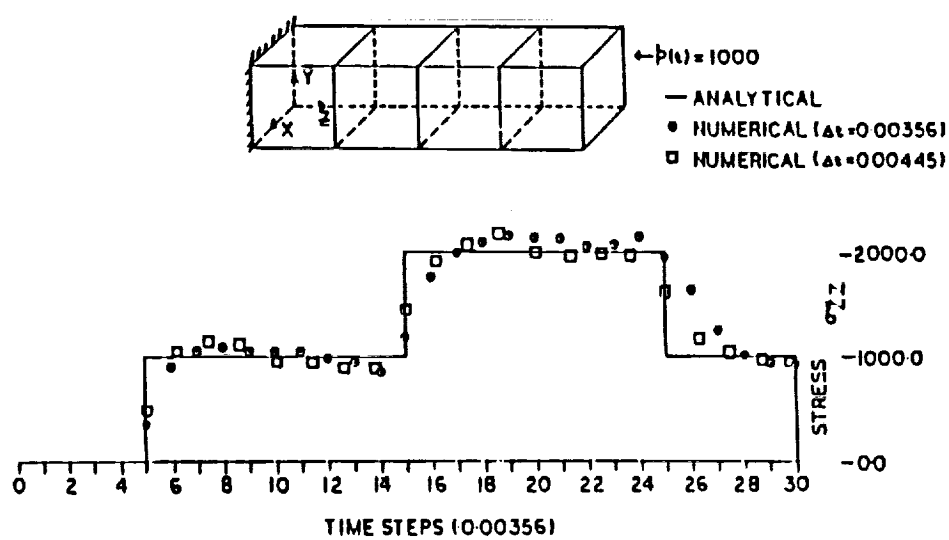


Fig. 7.4.2 Longitudinal stress at the mid-length of a column subjected to a suddenly applied end load

Dynamic analysis of a buried foundation [61]:

A number of meshes used are shown in Figure 7.4.3a,b and the results are compared with the exact analytical solution and previous BEM results in Figure 7.4.3c. It can be seen from Figure 7.4.3c that the results from the present analysis agree very well with the analytical and finite element results. At this point, it is important to mention that in the FEM analysis 1000 axisymmetric elements were used to obtain the correct results, whereas in the present BEM analysis the problem is modeled in three-dimensions with only 16 surface elements.

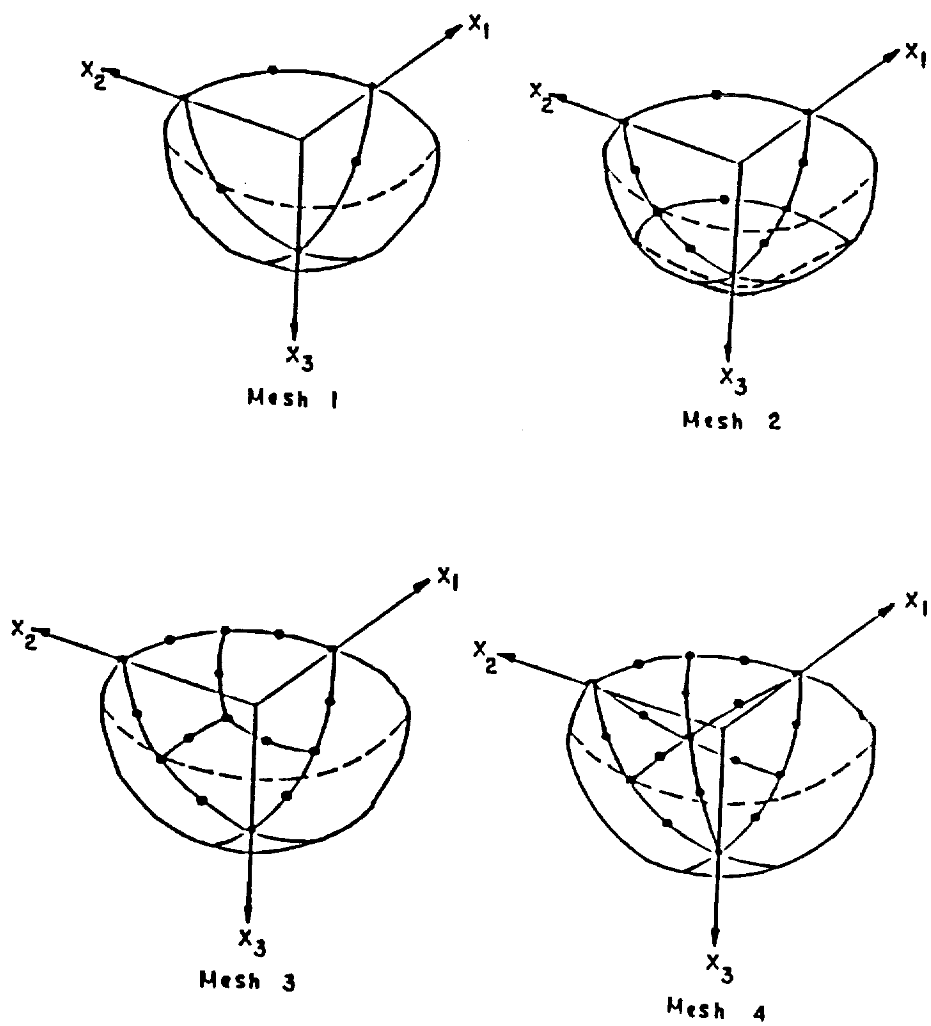


Fig. 7.4.3a Discretizations of hemispherical foundation-soil interfaces

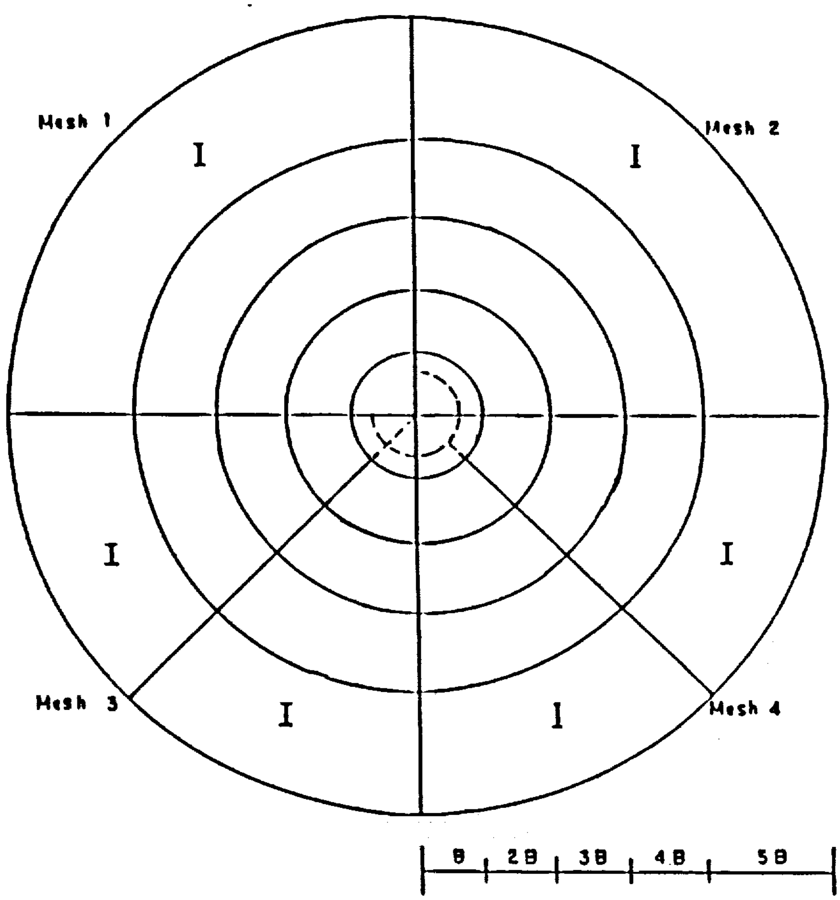


Fig. 7.4.3b Discretizations of the free field for a hemispherical foundation

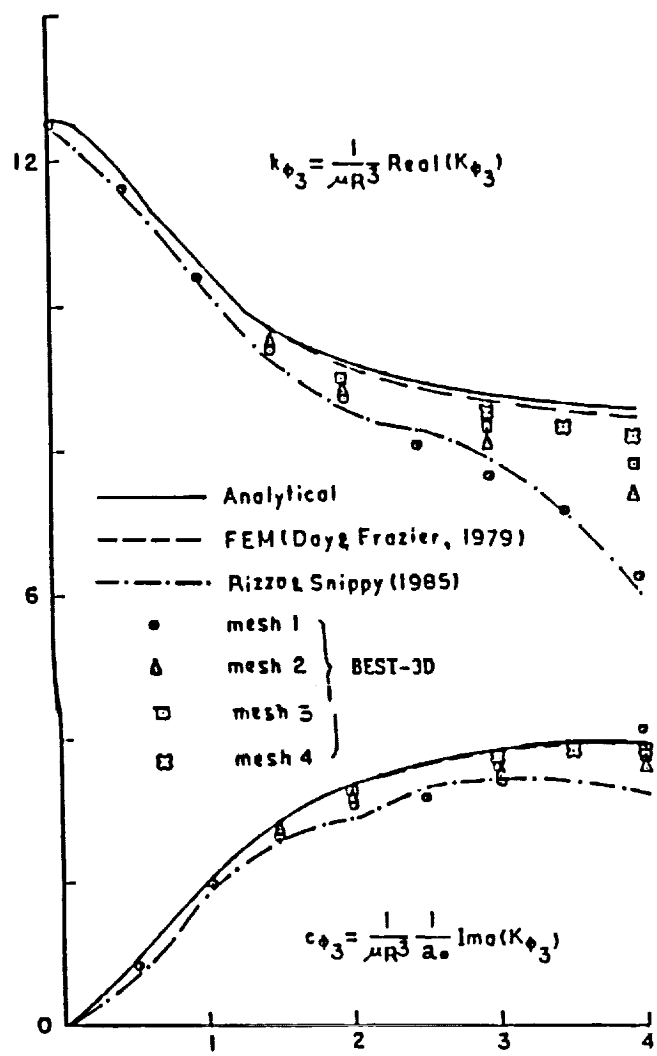


Fig. 7.4.3c Comparison of torsional stiffness coefficients for a hemispherical foundation

Application to the vibration isolation problem [41]:

In this example, results using **BEST3D** for a number of vibration isolation cases are compared with the actual field observation of Woods (62). He carried out a series of field tests in his attempt to define the screened zone and degree of amplitude reduction within the screened zone for trenches of a few specific shapes and sizes. He classified the foundation isolation problem into two categories, namely active isolation and passive isolation. Active isolation is the employment of barriers close to or surrounding the source of vibrations to reduce the amount of wave energy radiated away from the source. Passive isolation is the employment of barriers at points remote from the source of vibrations but near a site where the amplitude of vibration must be reduced. These tests were performed at a selected site of a two-layer system as shown on Figure 7.4.4a. A layer of uniform silty fine sand (SM) with dry density $\rho = 104$ lb/ft^3, water content w = 7%, void ratio e = 0.61, pressure wave velocity $v_p = 940$ ft/sec, and the Rayleigh wave velocity $v_R = 413$ ft/sec rests on a layer of sandy silt (ML) with dry density $\rho = 91$ lb/ft^3, water content w = 23 %, void ratio e = 0.68, and pressure wave velocity $v_p = 1750$ ft/sec. The field tests for the active isolation were carried out for four lengths of the trench forming an arc of 90, 180, 270, and 360 degrees. The depth of the trench was varied between 0.5, 1.0, and 2.0 feet, while the width was 0.25 feet for all trenches. The distance from the center of footing to the centerline of trench is 1.0 foot. The field tests for the passive isolation were carried out for six lengths of trench, namely 1.0, 2.0, 3.0, 4.0, 6.0, and 8.0 feet. Distances from the center of footing to the centerline of trench were 5 and 10 feet and depths of trench were 1.0, 2.0, 3.0, and 4.0 feet. A constant input excitation force vector of 18 lb was used in all active and passive tests. The applied operational frequencies of the machine were 200, 250, 300, and 350 Hertz.

In the present investigation, the soil profile is modeled as a two-layer system defined by the shear modulus G, Poisson's ratio ν, and mass density ρ. The material properties of the top layer are basically determined by the relations among the pressure wave velocity, the shear wave velocity, and the Rayleigh wave velocity. Accordingly, the values are taken as shear modulus G = 647200 lb/ft^2, Poisson's ratio $\nu = 0.35$, and the mass density $\rho = 3.25$ lb-sec/ft^3. The material properties of the bottom layer are determined by assuming the Poisson's ratio to be the same as that of the top layer. Accordingly, the values are taken as shear modulus G = 1991150 lb/ft^2, Poisson's ratio $\nu = 0.35$, and the mass density $\rho =$ 2.84 lb-sec/ft^3. The diameter of the vibration excitor which was not specified, is assumed to be 0.50 feet. It was thought that this dimension has relatively less influence on the results. A distributed pressure of 81.5 lb/ft^2 which is equivalent to 18 lb force is applied to the footing.

For the active isolation problem, a typical comparison of the BEM solutions and the experimental results are shown in Figures 7.4.4b and 7.4.4c. Figure 7.4.4b shows the overview of a contour diagram for a half circle trench with depth of 1.0 foot under an operating frequency of 250 Hertz. Both approaches predict similar screened zones which are area symmetrical about a radius from the source of excitation through the center of the trench and bounded laterally by two radial lines extending from the center of the source

of excitation through points 45 degrees from each end of the trench. The expanding of screened zones may be noted when the screened zones in Figure 7.4.4c, which is obtained by extending the length of trench from half circle to 270 degrees, are compared with those of Figure 7.4.4b.

For the passive isolation problem, comparisons of the BEM solutions and the experimental results are shown in Figure 7.4.4d which shows the contour diagram of amplitude reduction factor for a rectangular trench of six feet in length and 2 feet in depth under an operating frequency of 250 Hertz. This trench is located 5 feet from the center of source. The comparison shows that again the experimental results are predicted qualitatively by the BEM solution. It should be noted that similar isolation analyses could also be carried out by **BEST3D** in the areas of transient dynamics as well as those occurring in the areas of periodic and transient acoustics.

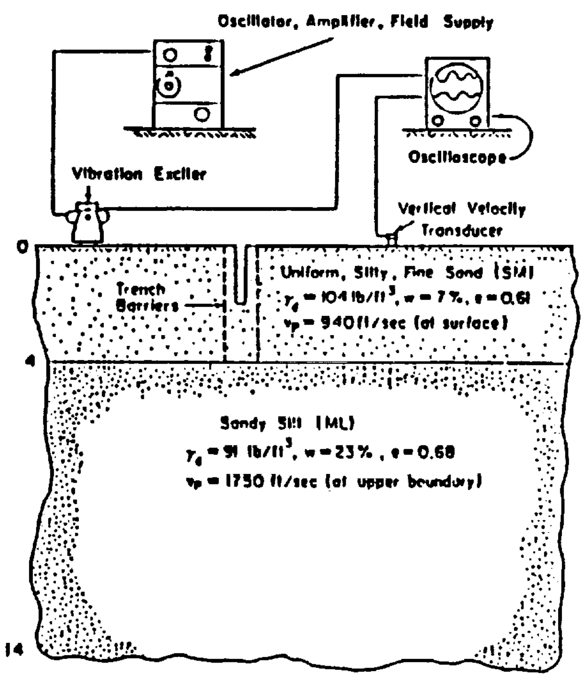


Fig. 7.4.4a Field site soil properties and schematic of instrumentation

ORIGINAL PAGE IS OF POOR QUALITY

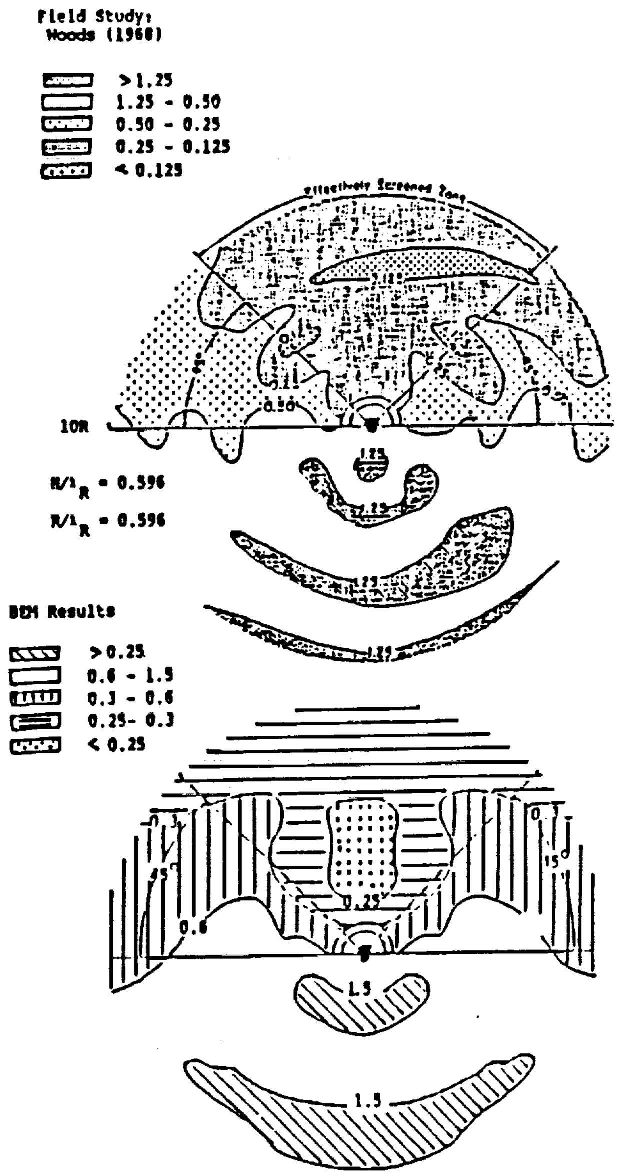


Fig. 7.4.4b Amplitude reduction factor contour diagram

ORIGINAL PAGE IS
OF POOR QUALITY

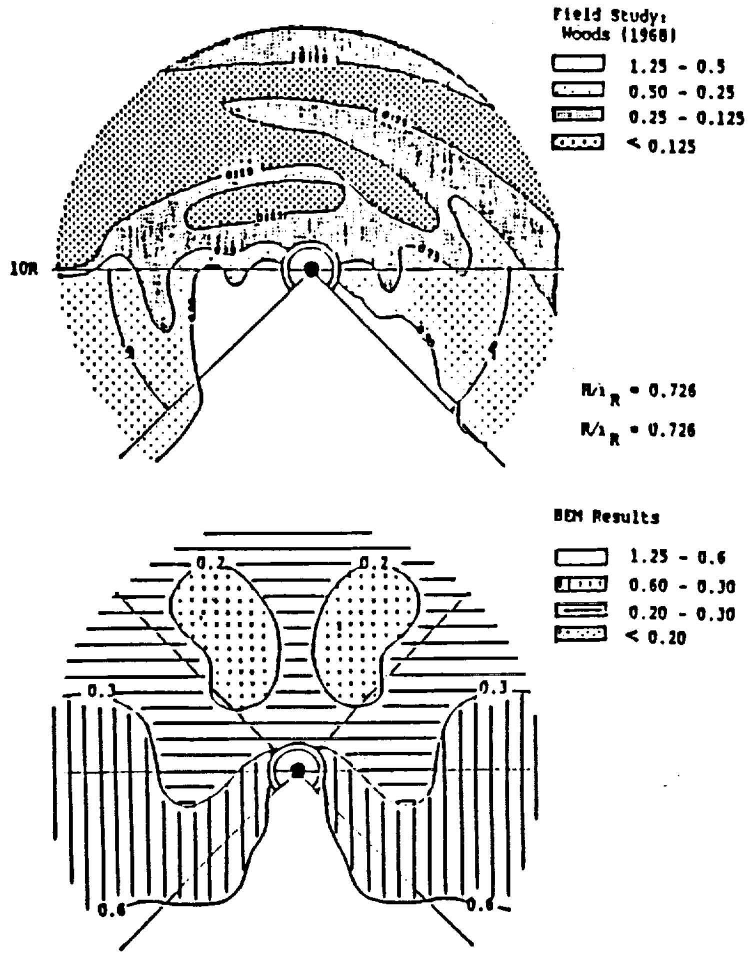


Fig. 7.4.4c Amplitude reduction factor contour diagram

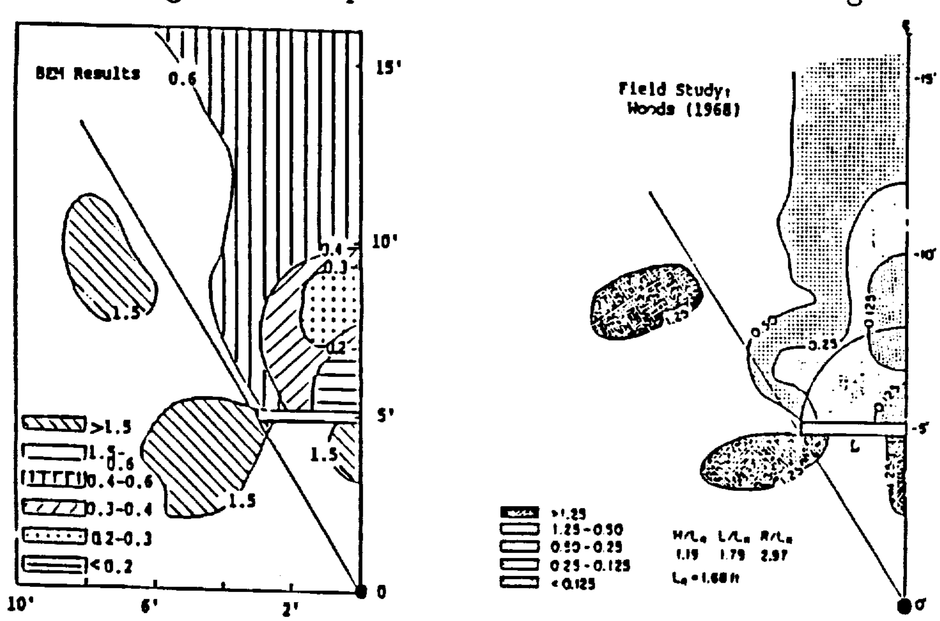


Fig. 7.4.4d Amplitude reduction factor contour diagram

ORIGINAL PAGE IS
OF POOR QUALITY

7.5 NON-LINEAR STRESS ANALYSIS

Benchmark Notch Specimen [34,36]:

The benchmark notch specimen is a double edge notch specimen developed by General Electric/Louisiana State University (GE/LSU) under NASA-Lewis Contract NAS3-22522. These data have been used to verify the elastic and inelastic capabilities of the present analysis.

Stress analysis was carried out for the gage section only, a procedure already known to be satisfactory. For the elastic analysis three different models were used, all idealizing one-quarter of the specimen gage section. Detailed comparison of results was carried out among the present analyses and finite element results. While these comparisons are not discussed here, it should be noted that, with sufficient mesh refinement, equivalent results were obtained with all analysis tools.

The discussion in this paper is directed at the comparison of present results with the GE/LSU strain gage data. The major characteristics of all of the analyses are summarized in the table below. The maximum peripheral strain in the notch (at the free surface) is given for each analysis. All analysis methods and the test data show that this value should be between 1700 and 1800 (microstrain).

Model	Displacement and Traction Elements	Variation	Equations	Subregions	CPU Time (Seconds)	Max Strain
1	50	linear	156	1	64	1688
2	50	quadratic	456	1	234	1780
3	22	linear	78	2	20	1594
4	22	quadratic	210	2	60	1742
5	22	mixed	117	2	31	1729
6	10	linear	36	1	10	1186
7	10	quadratic	96	1	28	1605

It is clear from the table that models 2, 4 and 5 all yield results of accuracy entirely comparable with the strain gage data. The variation in peak strain among these three is within $\pm 1.5\%$. It is also clear that the most cost effective analysis is that which combined substructuring with mixed linear and quadratic variation.

The first nonlinear analysis carried out for the specimen was a monotonic loading from zero load to the maximum load used in the test program. For this analysis quadratic variation was used for all surface elements. It was observed that the results gave generally good agreement with the experimental data, but that the overall strain level was somewhat too low. The analysis was repeated using a weighted surface model (Figure 7.5.1a) to capture better the gradients near the notch.

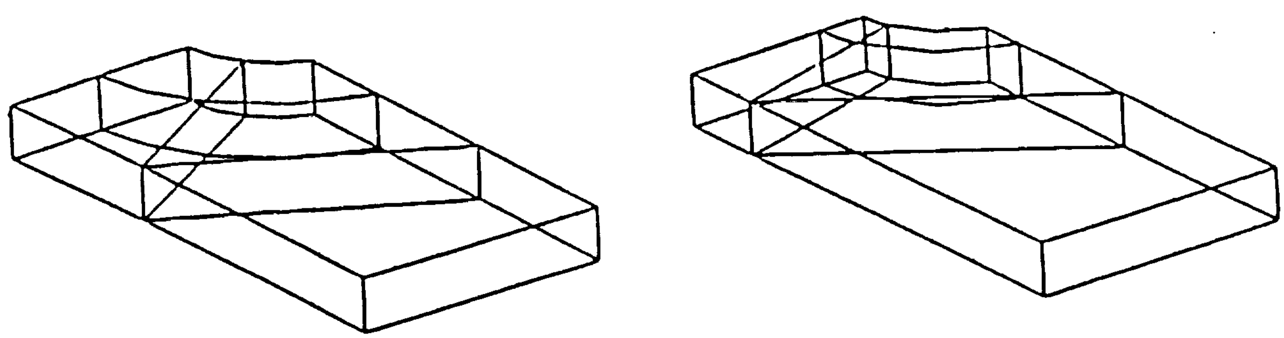


Fig. 7.5.1a Regular and Weighted Boundary Element Surface Model for Benchmark Notch

The weighted model was then used for the cyclic analysis of the specimen. The results of this analysis (Figure 7.5.1b) show excellent agreement with the measured notch root strains.

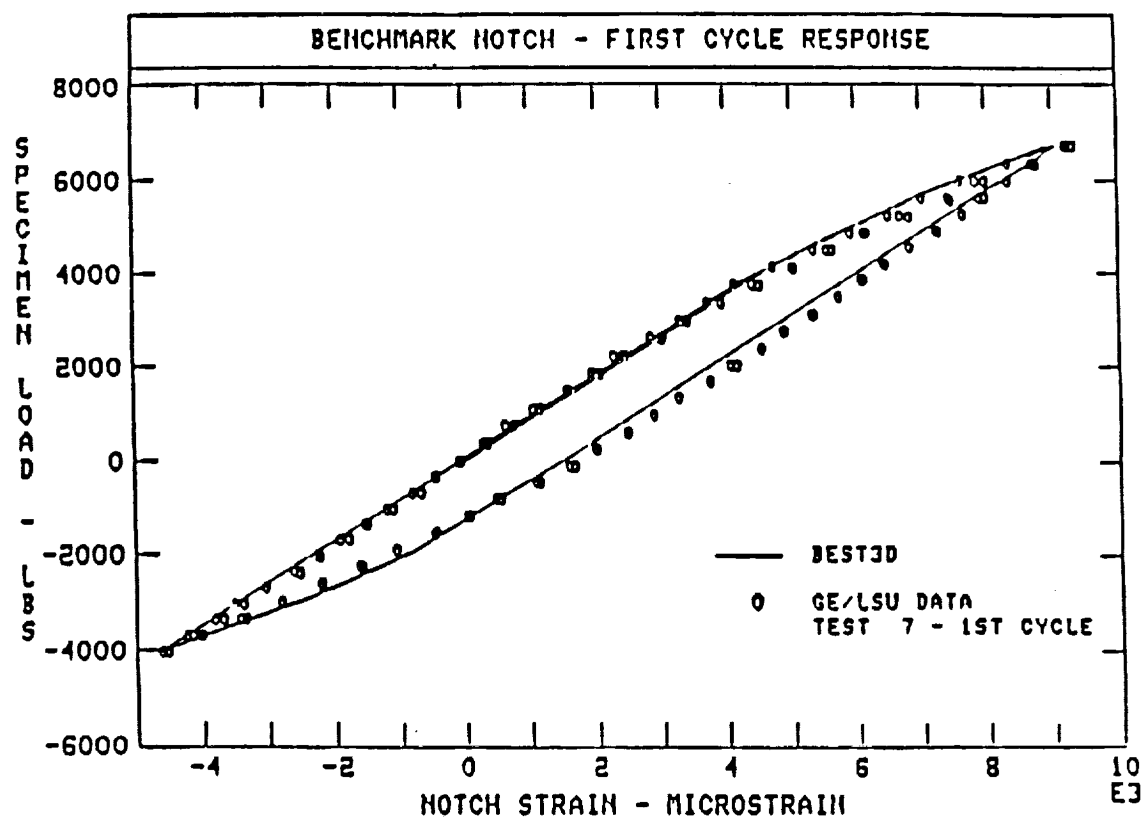


Fig. 7.5.1b Cyclic Inelastic Notch Response

Three-dimensional Analysis of a Notch Plate [43,44,45]:

The plastic deformation of a notch plate subjected to tension is analyzed under plane stress conditions using both the iterative and variable stiffness algorithms. The material properties for the plate are:

$$E = 7000 \text{ kg/mm}^2$$

$$\nu = 0.2$$

$$\sigma_o = 24.3 \text{ kg/mm}^2 \text{ (Von Mises yield criterion)}$$

$$h = 0.0$$

A 90 degree notch is cut out of the sides of the plate. The maximum to minimum width ratio is 2 and the thickness is 3/30 of the maximum width. A quarter of the plate is discretized in two subregions as shown in figure 7.5.2a. The region, containing the notch, has 30 quadratic boundary elements. The inset in this figure shows 6 (twenty-node) isoparametric cells (with 68 distinct cell nodes) which are used in volume integration. The second region has 16 quadratic boundary elements. No volume discretization is required in this region since it remains elastic throughout the analysis. Boundary conditions on the front and back faces are assumed traction free.

In figure 7.5.2b, the stress-strain response on the mid-plane of the root is given for the two (3-D) BEM analyses and compared with the two-dimensional plane stress and plane strain BEM solutions obtained by Banerjee and Raveendra [25]. The three-dimensional results are in good agreement with one another, and fall between the two-dimensional solutions, closer to the plane stress result, as one would expect. Also shown here are the results of particular integral analysis.

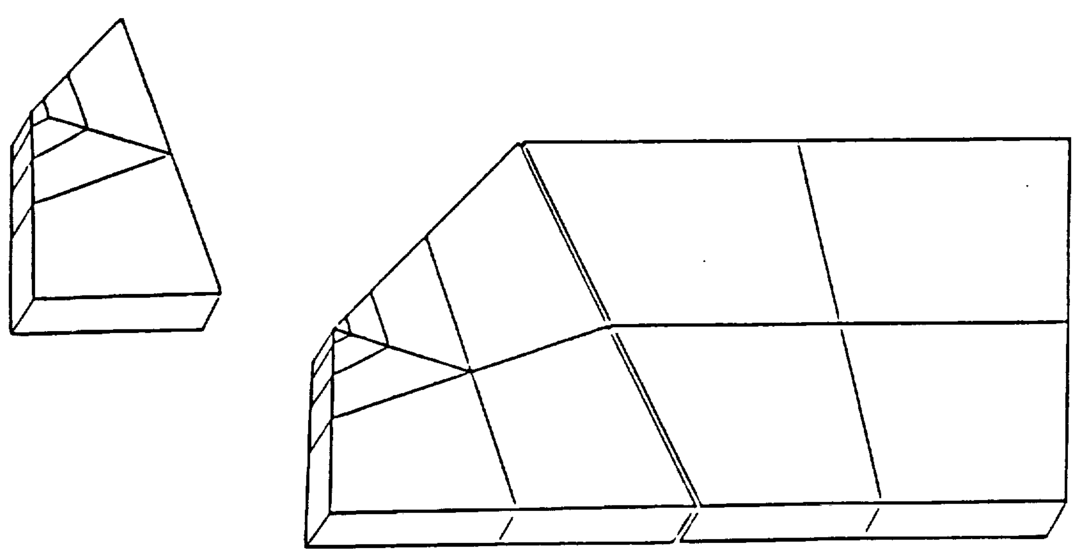


Fig. 7.5.2a Boundary and volume discretization of a three-dimensional notch plate

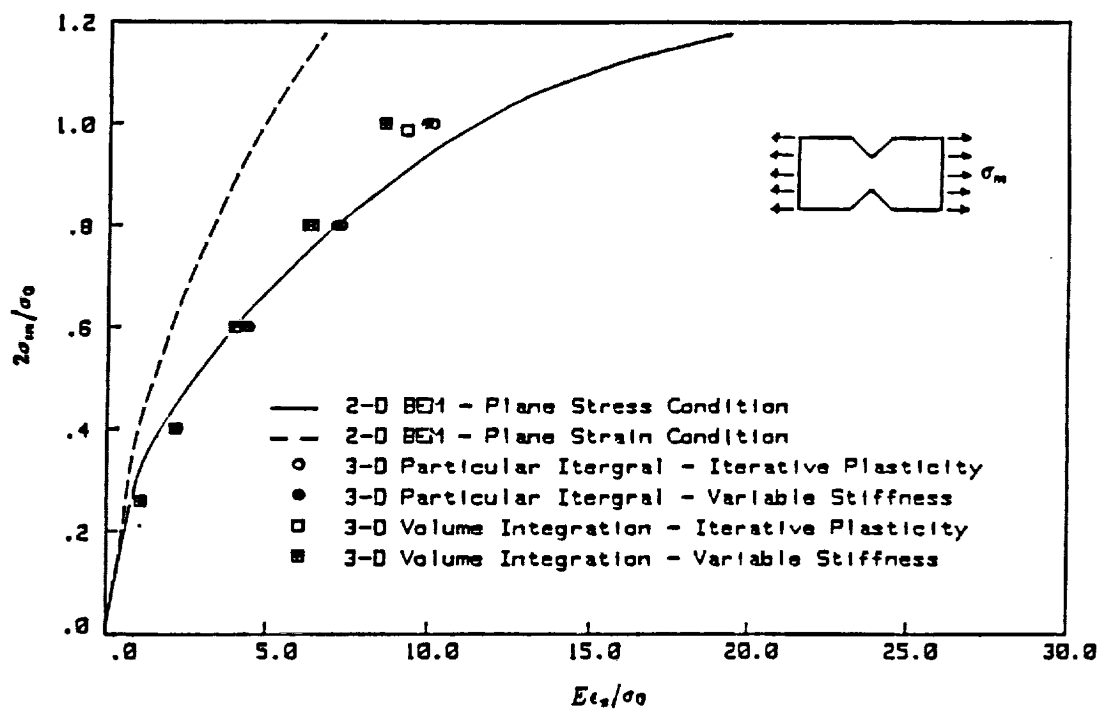


Fig. 7.5.2b Stress-strain response (on the mid-plane) at the root of a three-dimensional notch plate

Three-dimensional Analysis of a Perforated plate[43,44,45]:

The plastic deformation of a perforated plate in tension is analyzed under plane stress conditions using both the iterative and the variable stiffness solution algorithms. The material properties for the plate are:

$E = 7000 \text{ kg/mm}^2$

$\nu = 0.2$

$\sigma_o = 24.3 \text{ kg/mm}^2$ (Von Mises yield criterion)

$h = 224.0$

The diameter of the circular hole, at the center of the plate, is one-half the width, and the thickness is one-fifth the width. A quarter of the plate is discretized in 2 subregions, as shown in figure 7.5.3a. The first region, containing the root of the plate has 30 quadratic bondary elements and the second region has 23 boundary elements. No volume discretization is required in this region since it remains elastic throughout the analysis. Boundary conditions on both the front and back faces are assumed traction free.

This problem was previously analyzed experimentally by Theocaris and Marketos and by Zienkiewicz [16] using the finite element method. The stress-strain response at the root of the plate is shown in figure 7.5.3b. The results obtained using the various BEM algorithms show good agreement with one another and with the variable stiffness FEM analysis. Differences between the iterative and variable stiffness BEM formulations are much less significant than the differences in the 2 FEM algorithms. In order to evaluate the degree of convergence of the results, a mesh with 16 volume cells was studied. The results were unchanged from those shown. Those obtained from the particular integrals are also shown here for comparison.

The present analysis was carried out on the Cray-1 computer. The CPU times for the two algorithms were: 272 seconds for iterative procedure and 361 seconds for the variable stiffness method.

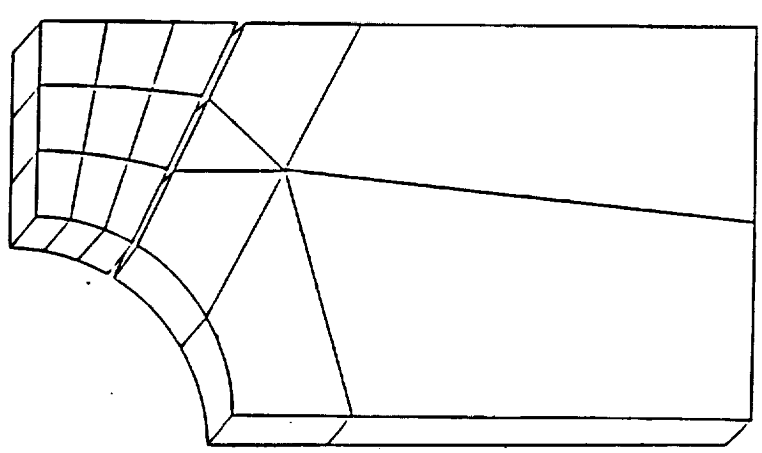


Fig. 7.5.3a Three-dimensional mesh of a perforated plate

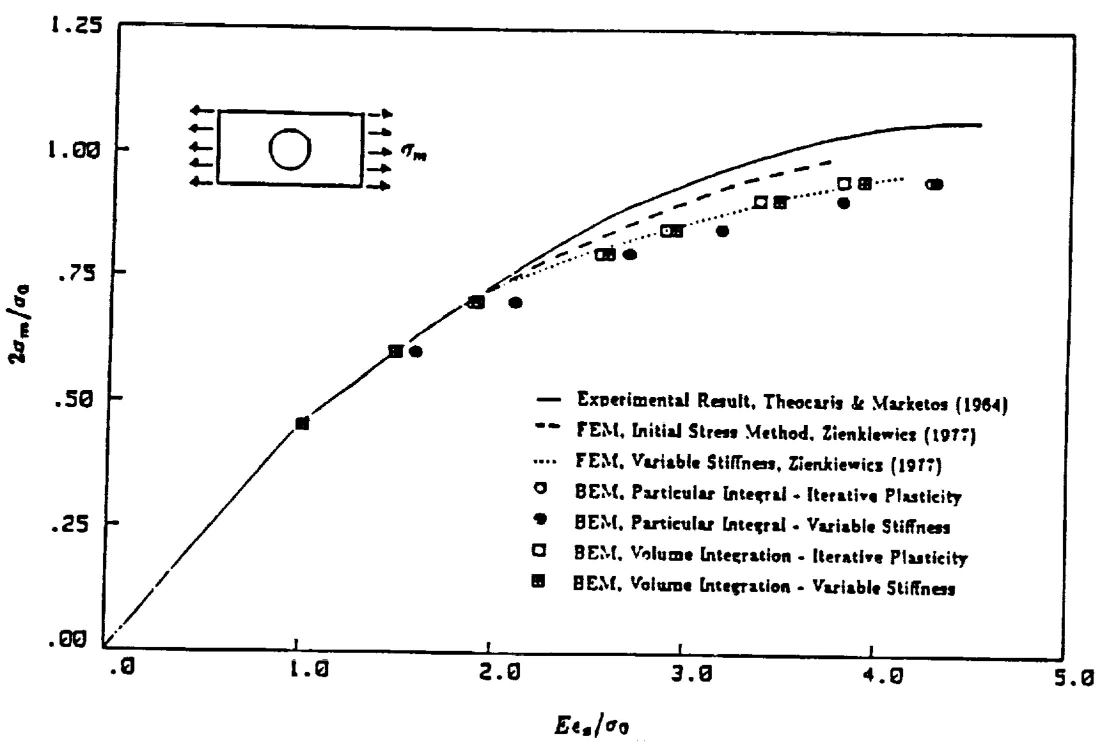


Fig. 7.5.3b Stress-strain response at the root of a three-dimensional perforated plate

8.0 REFERENCES

1. BANERJEE, P.K., and BUTTERFIELD, R., (1975), Boundary Element Method in Geomechanics, Chapter 16 in Finite Element in Geomechanics, Ed. G. Gudehus, John Wiley.
2. BANERJEE, P.K., and BUTTERFIELD, R., (1981), Boundary Element Methods in Engineering Science, McGraw-Hill, London.
3. BREBBIA, C.A., and WALKER, S., (1980), Boundary Element Techniques in Engineering, Newness-Butterworths, London.
4. BREBBIA, C.A., TELLES, J.C.F., and WROBEL, L.C., (1984), Boundary Element Techniques, Springer-Verlag.
5. LIGGETT, J., and LIU, P., (1983), Boundary Integral Equation for Porous Media Flow, Allen and Unwin, London.
6. CROUCH, S.L., and STARFIELD, A.M., (1983), Boundary Element Methods in Solid Mechanics, Allen and Unwin, London.
7. MUKHERJEE, S., (1982), Boundary Element Methods in Creep and Fracture, Applied Science Publishers, London.
8. BANERJEE, P.K., and BUTTERFIELD, R., Editors, (1979), Developments in Boundary Element Methods I, Elsevier Applied Science Publishers, Barking, Essex, U.K.
9. BANERJEE, P.K., and SHAW, R.P., Editors, (1982), Developments in Boundary Element Methods II, Elsevier Applied Publishers, Barking, Essex, U.K.
10. BANERJEE, P.K., and MUKHERJEE, S., Editors, (1984), Developments in Boundary Element Methods III, Elsevier Applied Science Publishers, Barking, Essex, U.K.
11. BANERJEE, P.K., and WATSON, J.O., Editors, (1986), Developments in Boundary Element Methods IV, Elsevier Applied Science Publishers, Barking Essex, U.K.
12. CRUSE, T.A., and RIZZO, F.J., Editors, (1975), Boundary Element Equation Method: Computational Applications in Applied Mechanics, AMD-Vol. 11, ASME, New York.
13. CRUSE, T.A., PIFKO, A., and ARMEN, H., Editors, (1985), Advanced Topics in Boundary Element Analysis, AMD-Vol. 72, ASME, New York.
14. RIZZO, F.J., and SHIPPY, D.J., (1977), An advanced boundary integral equation method for three-dimensional thermo-elasticity, Int. J. Num. Meth. in Engng., 11, 1753- 1768.

15. RIZZO, F.J., and SHIPPY, D.J., (1979), Recent advances of the boundary element method in thermoelasticity, Chapter VI in Banerjee, P.K. and Butterfield, R. (Eds.) Developments in Boundary Element Methods I, Elsevier Applied Science Publishers, Barking, Essex, U.K.

16. ZIENKIEWICZ, O.C., (1977) Finite Element Method in Engineering Science, McGraw Hill, London.

17. CRUSE, T.A., and WILSON, R.B., (1978), Boundary-integral equation methods for elastic fracture mechanics analysis, AFOSR-TR-77-0355.

18. BANERJEE, P.K., CATHIE, D.N., and DAVIES, T.G., (1979), Two and three dimensional problems of elastoplasticity, in Developments in Boundary Element Methods I, Banerjee, P.K. and Butterfield, R. (Eds.), Elsevier Applied Science Publishers, Barking, Essex, U.K.

19. BANERJEE, P.K., and CATHIE, D.N., (1980), A direct formulation and numerical implementation of the boundary element method for two-dimensional problems of elastoplasticity, Int. J. Mech. Sci., 22, 233-245.

20. CATHIE, D.N., and BANERJEE, P.K., (1980), Boundary element methods in axisymmetric plasticity, Innovative Numerical Analysis for the Applied Engineering Sciences, Shaw, R.P., et al (Eds.), University of Virginia Press.

21. CATHIE, D.N., and BANERJEE, P.K., (1982), Boundary element methods for plasticity and creep including a viscoplastic approach, Res. Mechanica, Vol. 4, 3-22.

22. KUMAR, V., and MUKHERJEE, S., (1977), A boundary-integral equation formulation for time-dependent inelastic deformation in metals, Int. J. Mech. Sci., 19, 713-724.

23. BANERJEE, P.K., and DAVIES, T.G., (1984), Advanced implementation of boundary element methods for three-dimensional problems of elastoplasticity and viscoplasticity, Chapter I in Developments in Boundary Element Methods III, Applied Science Publishers, London.

24. BANERJEE, P.K., and WILSON, R.B., (1984), Fundamental solutions of the elasticity equations for distributed hot spots and cavities, Unpublished work.

25. BANERJEE, P.K., and RAVEENDRA, S.T., (1986), Advanced boundary element method for two and three-dimensional problems of elastoplasticity, Int. J. Num. Meth. in Engng, Vol. 23, 985-1002.

26. BANERJEE, P.K., and AHMAD, S., (1985), Advanced three- dimensional dynamic analysis by boundary element methods, AMD-Vol. 72, 65-82.

27. BANERJEE, P.K., AHMAD, S., and MANOLIS, G.D., (1986), Transient elastodynamic analysis of three-dimensional problems by boundary element method, Earthquake Engng. and Struct. Dynamics, Vol. 14, 933-949.

28. AHMAD, S., and BANERJEE, P.K., (1988), Time domain transient elastodynamic analysis of 3-D solids by BEM, Int. Jour. Numerical Methods in Engg., Vol. 26, 1709-1728.

29. AHMAD, S., and BANERJEE, P.K., (1986), Free-Vibration analysis by BEM using particular integrals, Journal of Engineering Mechanics, ASCE, Vol. 112, No. 7, 682-695.

30. MANOLIS, G.D., and BANERJEE, P.K., (1986), Conforming and non-conforming boundary elements in 3-D elastostatics, Int. J. Num. Meth. in Engng, Vol. 23, 1885-1904.

31. LACHAT, J.C., and WATSON, J.O., (1976), Effective numerical treatment of boundary integral equations: A formulation for three-dimensional elastostatics, Int. J. Num. Meth. in Engng., 10, 991-1005.

32. WATSON, J.O., (1979), Advanced implementation of the boundary element method for two and three-dimensional elastostatics, in Banerjee, P.K. and Butterfield, R. (Eds.) Developments in Boundary Element Methods I, Elsevier Applied Science Publishers, Barking Essex, U.K., 31-64.

33. BANERJEE, P.K., WILSON, R.B., and MILLER, N., (1985), Development of a large BEM system for three-dimensional inelastic analysis, AMD-Vol. 72, Edited by Cruse, T.A., Pifko, A., and Armen, H., 1-20.

34. WILSON, R.B., SNOW, D.W., and BANERJEE, P.K., (1985), Stress analysis of gas turbine engine structures using boundary element methods, AMD-Vol. 72, 45-64.

35. BANERJEE, P.K., WILSON, R.B., and MILLER, N., (1988), Advanced elastic and inelastic stress analysis of gas turbine engine structures by BEM, Int. Jour. Num. Methods in Engg., Vol. 26, 393-411.

36. BANERJEE, P.K., WILSON, R.B., and RAVEENDRA, S.T., (1987), Advanced applications of BEM to three-dimensional problems of monotonic and cyclic plasticity, International Jour. Mechanical Sciences, Vol. 29, No.9, 637-653.

37. BANERJEE, P.K. and RAVEENDRA, S.T., (1987), A new boundary element formulation for two-dimensional elastoplastic analysis, Engineering Mechanics Division, ASCE, Vol. 113, No.2, 252-265.

38. HENRY, D., PAPE, D. and BANERJEE, P.K., (1987), New axisymmetric BEM formulation for body forces using particular integrals, Engineering Mechanics Division, ASCE, Vol. 113, No.5, 671-688.

39. HENRY, D. and BANERJEE, P.K., (1987), A thermoplastic BEM analysis for substructured axisymmetric bodies, Jour. Engineering Mechanics Division, Vol. 113, No. 12, 1880-1900.

40. HENRY, D. and BANERJEE, P.K., (1988), A variable stiffness type boundary element formulation for axisymmetric elastoplastic media, Int. Jour. Num. Methods in Engg., Vol. 26, 1005-1027.

41. BANERJEE, P.K., AHMAD, S. and CHEN, K. (1987), Advanced application of BEM to wave barriers in multi-layered three-dimensional soil media, Earthquake Engineering and Structural Dynamics, Vol. 16, 1041-1060.

42. DARGUSH, G. F. and BANERJEE, P.K. (1989), Advanced BEM for steady state heat tranfer analysis, Int. Jour. Num. Methods in Engineering, Vol. 28, 2123-2142.

43. BANERJEE, P.K., HENRY, D. and RAVEENDRA, S.T. (1989), Advanced BEM for inelastic analysis of solids, Int. Jour. Mechanical Sciences, Vol. 31, 309-322.

44. HENRY, D. and BANERJEE, P.K. (1988), A new boundary element formulation for two and three-dimensional problems of thermoelasticity using particular integrals, Int. Jour. Num. methods in Engineering, Vol. 26, 2061-2077.

45. HENRY, D. and BANERJEE, P.K. (1988), A new boundary element formulation for two and three-dimensional problems of elastoplasticity using particular integrals, Int. Jour. Num. methods in Engineering, Vol. 26, 2079-2096.

46. MROZ, Z. (1967) On the description of anisotropic work hardening, Jour. Mech. Phys. Solids, Vol. 15, 163-175.

47. KRIEG, R.D. (1975) A practical two surface plasticity theory, Jour. Appl. Mech. Trans. ASME, Vol. E42, 641-646.

48. DAFALIAS, Y.F. and POPOV, E.P. (1974) A model of nonlinearly hardening material for complex loadings, Proc. 7th U.S. National Congress in applied Mech., Boulder. Acta Mech., Vol. 21, 173-192.

49. WALKER, K.P. (1981) Research and Development Program for Nonlinear Structural Modeling with Advanced Time-Temperature Dependent Constitutive Relationships, NASA CR-165533.

50. CASSENTI, B.N. and THOMPSON, R.L. (1983) Material Response Predictions for Hot Section Gas Turbine Engine Components, AIAA-83-2020, presented at the AIAA/SAE/ASME 19th Joint Propulsion Conference, Seattle, Washington, June 27-29.

51. AHMAD, S. and BANERJEE, P.K. (1988), Multi-domain BEM for two-dimensional problems of elastodynamics, Int. Jour. Num. Methods in Engg., Vol. 26, 891-911.

52. PAPE, D. and BANERJEE, P.K. (1987), Treatment of body forces in 2D elastostatic BEM using particular integrals, Jour. Applied Mechanics, Vol. 54, 871-886.

53. WILSON, R.B., MILLER, N.M. and BANERJEE, P.K. (1990), Calculations of Natural Frequencies and Mode shapes for three-dimensional Solids by BEM, Int. Jour. Numerical Methods in Engineering, Vol. 29, 1737-1757.

54. BANERJEE, P.K. and HENRY, D. (1991), Analysis of three-dimensional solids with holes by BEM, to appear in Int. Jour. Num. Methods in Engineering.

55. BANERJEE, P.K., HENRY, D. and DARGUSH, G. (1989), Analysis of 3-Dimensional Solids with Multiple Inserts by BEM, submitted to Int. Jour. Num. Methods in Engineering.

56. LEISSA, A. and ZHANG, Z. (1983), On the three-dimensional vibrations of the cantilevered rectangular parallelepiped, Jour. of the Acoustical Soc. of America., Vol. 73, No. 6, 2013-2021.

57. MacBAIN, J.C. and KIELB, R.E., and LEISSA, A.W. (1984), Vibrations of twisted cantilevered plates - experimental investigation, ASME Paper 84-GT-96.

58. KIELB, R.E., LEISSA, A.W., MacBAIN, J.C. and CARNEY, K.S. (1985), Joint Research Effort on Vibrations of Twisted Plates - Phase I: Final Results, NASA Reference Publication No. 1150.

59. DARGUSH, G. and BANERJEE, P.K. (1991), Advanced BEM for transient heat transfer, to appear in Int. Jour. Num. Methods in Engineering.

60. CARLSLAW, H.S. and JAEGER, J.C. (1947), Conduction of Heat in Solids, Clarendon Press, Oxford.

61. ISRAIL, A.S.M. and BANERJEE, P.K. (1990), Effects of geometrical and material properties on the vertical vibration of 3-D foundations by BEM, Int. Jour. Num. Anal. Methods in Geomechanics, Vol. 14, 49-70.

62. WOODS, R. D. (1968), Screening of Surface Waves in Soils, J. Soil Mech. and Found. Div., Proc. ASCE, Vol. 94, SM 4, July, pp. 951-979.

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No. NASA CR-185298	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle BEST3D User's Manual Boundary Element Solution Technology, 3-Dimensional; Version 3.0		5. Report Date March 1991 6. Performing Organization Code
7. Author(s) P. K. Banerjee and R. B. Wilson		8. Performing Organization Report No. PWA-5940-74 10. Work Unit No.
9. Performance Organization Name and Address United Technologies Corporation Pratt & Whitney 400 Main Street East Hartford, Connecticut 06108		11. Contract or Grant No. NAS3-23697
12. Sponsoring Agency Name and Address NASA-Lewis Research Center 21000 Brookpark Road Cleveland, Ohio 44135		13. Type of Report and Period Covered Final Report 14. Sponsoring Agency Code

15. Supplementary Notes
NASA Project Manager, Mr. Dale A. Hopkins
21000 Brookpark Road, MS 49-8
Cleveland, Ohio 44135

16. Abstract

This report documents the theoretical basis and programming strategy utilized in the construction of the computer program BEST3D (Boundary Element Solution Technology - Three Dimensional) and provides detailed input instructions for the use of the program. An extensive set of test cases and sample problems is included in the manual and is also available for distribution with the program. The BEST3D program was developed under the 3-D Inelastic Analysis Methods for Hot Section Components contract (NAS3-23697). The overall objective of this program was the development of new computer programs allowing more accurate and efficient three-dimensional thermal and stress analysis of hot section components, i.e., combustor liners, turbine blades, and turbine vanes. The BEST3D program allows both linear and nonlinear analysis of static and quasi-static elastic problems and transient dynamic analysis for elastic problems. Calculation of elastic natural frequencies and mode shapes is also provided.

17. Key Words (Suggested by Author(s)) 3-D Inelastic Analysis, Finite Elements, Boundary Elements, High Temperature, Vibration, Solution Methods, Constitutive Modeling, Dynamics	18. Distribution Statement General Release

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price
Unclassified	Unclassified	436	A19

NASA FORM 1626 OCT 86